Integral Flow Modelling Approach for Surface Water-Groundwater Interactions along a Rippled Streambed

Tabea Broecker 1,*, Katharina Teuber 1, Vahid Sobhi Gollo 1, Gunnar Nützmann 2,3, Jörg Lewandowski 2,3 and Reinhard Hinkelmann 1

1 Chair of Water Resources Management and Modeling of Hydrosystems, Technische Universität Berlin, 13355 Berlin, Germany
2 Ecohydrology Department, Leibniz-Institute of Freshwater Ecology and Inland Fisheries, 12587 Berlin, Germany
3 Geography Department, Humboldt-University Berlin, 10099 Berlin, Germany
* Correspondence: tabea.broecker@uwi.tu-berlin.de; Tel.: +30-314-72-444

Received: 18 June 2019; Accepted: 16 July 2019; Published: 22 July 2019

Abstract: Exchange processes of surface and groundwater are important for the management of water quantity and quality as well as for the ecological functioning. In contrast to most numerical simulations using coupled models to investigate these processes, we present a novel integral formulation for the sediment-water-interface. The computational fluid dynamics (CFD) model OpenFOAM was used to solve an extended version of the three-dimensional Navier–Stokes equations which is also applicable in non-Darcy-flow layers. Simulations were conducted to determine the influence of ripple morphologies and surface hydraulics on the flow processes within the hyporheic zone for a sandy and for a gravel sediment. In- and outflowing exchange fluxes along a ripple were determined for each case. The results indicate that larger grain size diameters, as well as ripple distances, increased hyporheic exchange fluxes significantly. For higher ripple dimensions, no clear relationship to hyporheic exchange was found. Larger ripple lengths decreased the hyporheic exchange fluxes due to less turbulence between the ripples. For all cases with sand, non-Darcy-flow was observed at an upper layer of the ripple, whereas for gravel non-Darcy-flow was recognized nearly down to the bottom boundary. Moreover, the sediment grain sizes influenced also the surface water flow significantly.

Keywords: groundwater-surface water interactions; integral model; computational fluid dynamics; hyporheic zone; OpenFOAM; ripples

1. Introduction

Hyporheic exchange—the exchange of stream and shallow subsurface water—is controlled by pressure gradients along the streambed surface and subsurface groundwater gradients. Over multiple scales, the bedform induced hyporheic exchange was identified as a crucial process for the biogeochemistry and ecology of rivers [1–10]. On large and intermediate scales, stream stage differences, meander loops or bars can generate hyporheic exchange. Accordingly, it is possible to control surface water-groundwater exchange by river stage manipulation e.g., to manage the inflow of saline groundwater into a river [11]. A decrease of the groundwater level, in turn, impacts surface water infiltration up to a maximum where groundwater and surface water are disconnected. This condition is achieved when the clogging layer does not cross the top of the capillary zone above the water table [12]. On small scales, river sediments usually form topographic features such as dunes or ripples. The flowing fluid encounters an uneven surface on the permeable streambed, which results in an irregular pattern in the pressure along that surface and induces hyporheic exchange [11–13].
Within theoretical, experimental, and computational studies the general mechanics of the bedform induced hyporheic exchange were examined over the past decades. By manipulating streambed morphology, stream discharge, and groundwater flow, experiments have been used to study driving forces for the hyporheic exchange intensively [14–17]. At submerged structures such as pool-riffle sequences or ripples, turbulences, eddies or hydraulic jumps may occur. Packman et al. [15], Tonina and Buffington [18], Voermans et al. [19] and other studies showed, that turbulence influences hyporheic exchange and should not be ignored. Facing these complex three-dimensional flow dynamics at the sediment-water interface, it can be challenging to establish suitable flume experiments or field studies. Computational fluid dynamics has proven to be a viable alternative. The majority of these studies have focused on surface-subsurface coupled models. Reasons for the application of different models for the surface and the subsurface are for example the strong temporal variability in streams including relatively high velocities, whereas the velocities and temporal variabilities in the groundwater are usually several orders of magnitude smaller, leading to different applied equations for the stream and the subsurface. Often, the two computational domains are linked by pressure. Pressure distributions from a surface water model are consequently used for a coupled groundwater model [20–26]. However, also fully coupled models such as the Integrated Hydrology Model [27] or HydroGeoSphere have already been successfully applied [28–30]. Within these models, open channel flow is described by the two-dimensional diffusion-wave approximation of the St. Venant equations, whereas the three-dimensional Richards equation is used for the subsurface. Water and solute exchange flux terms enable to simultaneously solve one system of equations for both flow regimes.

For many coupled surface-subsurface models, the Darcy law is applied within the sediment. However, especially for coarse bed rivers, this law may cause errors in the presence of non-Darcy hyporheic flow [15]. Following Bear [31], the linear assumption of the Darcy law is only valid if the Reynolds number does not exceed a value between 1 and 10. Applying Darcy’s law in non-Darcy-flow areas leads to an overestimating of groundwater flow rates [32]. Packman et al. [15] investigated hyporheic exchange through gravel beds with dune-like morphologies and applied the modified Elliot and Brooks model [33]. They realized that the model did not perform well—among other reasons—due to non-Darcy flow in the near-surface sediment which was not considered in the model. One possible solution to model groundwater in non-Darcy-flow areas is e.g., to use the Darcy-Brinkmann equation instead of the Darcy law. However, there is an additional parameter—the effective viscosity—which has to be determined.

In the present study, an extended version of the three-dimensional Navier–Stokes equations after Oxtoby et al. [34] is used for the whole system comprising the stream as well as the subsurface. For the application in the groundwater, sediment porosity, as well as an additional drag term, are included into the Navier–Stokes equations. The model is consequently also applicable for high Reynolds numbers within the subsurface where the Darcy law cannot be applied. To our knowledge, this solver was never used for the hyporheic zone before. We apply the new integral solver to evaluate the effect of ripple geometries and surface hydraulics on hyporheic exchange processes, based on the study by Broecker et al. [35] who investigated free surface flow and tracer retention over streambeds and ripples without considering the subsurface. In Broecker et al. [35] the three-dimensional Navier–Stokes equations were solved in combination with an implemented transport equation. In that study, ripple sizes, spacing as well as flow velocities affected pressure gradients and tracer retention considerably. Seven simulation cases were examined varying ripple height, length, distance, and flow rate. The investigated ripple geometries and flow rates are mainly transferred to the present study. Only case 6 is not used for the present study, as the irregular distance between the ripples gave no significant new findings compared to equal distances [35]. In contrast to Broecker et al. [35], the present study examines both free surface flow and subsurface flow. The aim of the present study is to evaluate the impact of ripple dimensions, lengths, spacing and surface velocity on flow dynamics within the hyporheic zone using a new integral model.
2. Materials and Methods

2.1. Geometry and Mesh

The analyzed geometry consists of a prismatic domain with a length of 15 m, a width of 1 m and a height of 1.5 m at the inlet and 1 m at the outlet. A weir structure is included in front of the outlet to fix the water level. A rippled area of approximately 3 m is introduced 6 m downstream of the inlet. The model geometry of the reference case with the corresponding initial water depth can be seen in Figure 1.

![Model geometry and initial condition for the water level (sediment: yellow, water: blue, air: gray); top: front view, bottom right: cross-section.](image)

The mesh has been discretized using the three-dimensional finite element mesh generator gmsh. The ripples have been generated using the three-dimensional tool called RippledField. The mesh is a mixture of the ported element and the ported profile of the ripples. The elements have been extruded with 10 layers in the y-direction to produce a three-dimensional mesh. Different meshes with similar mesh conditions have been created for three different models. The mesh resolution has been examined by calculating the fraction of the turbulent kinetic energy in the resolved motions after Pope [36], who suggests to resolve 80% of the turbulent kinetic energy for a well-resolved large eddy simulation. The mesh resolution has been chosen by analyzing the energy of the resolved motions and the energy of the unresolved motions. The resolutions have been validated for the air-phase and water-phase, which are of interest for our simulations. The air-phase submodel is detailed to simulate air-water interactions, which also show a significant effect on the pressure distribution at the streambed [31].

Table 1. Simulation cases including ripple geometries and flow rates.

<table>
<thead>
<tr>
<th>Case</th>
<th>Reference Case</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>ripple height (cm)</td>
<td>5.6</td>
<td>5.6</td>
<td>5.6</td>
<td>5.6</td>
<td>5.6</td>
<td>5.6</td>
<td></td>
</tr>
<tr>
<td>ripple length (cm)</td>
<td>20</td>
<td>12</td>
<td>17</td>
<td>20</td>
<td>5.6</td>
<td>5.6</td>
<td></td>
</tr>
<tr>
<td>flow rate (m/s)</td>
<td>0.25</td>
<td>0.25</td>
<td>0.25</td>
<td>0.25</td>
<td>0.25</td>
<td>0.25</td>
<td></td>
</tr>
<tr>
<td>ripple distance (cm)</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

2.2. Numerical Model

To simulate exchange processes of surface water and groundwater, the open source software Open Source Field Operation and Manipulation (OpenFOAM) version 2.4.0 has been used. A solver called "porousInter" has been applied. This solver was developed by Oxtoby et al. [34] and is based on the Open Source Field Operation and Manipulation (OpenFOAM) version 2.4.0. A solver called "porousInter" has been applied. This solver was developed by Oxtoby et al. [34] and is based...
interFoam solver by OpenFOAM. PorousInter is a multiphase solver for immiscible fluids and extends the three-dimensional Navier–Stokes equations by the consideration of soil porosity and effective grain size diameter. For our simulations two phases—water and air—are considered to allow water level fluctuations. Since the porousInter–solver does not account for the solid fraction of the soil, values that are represented by \( f \) are averaged only over the pore space volume. The conservation of mass and momentum are defined after Oxtoby et al. [34] as:

Mass conservation equation
\[
\varphi \nabla \cdot [\vec{U}^f] = 0
\]  
(1)

Momentum conservation equation
\[
\varphi \left( \frac{\partial [\rho^f \vec{U}^f]}{\partial t} + \nabla \cdot ([\rho^f \vec{U}^f] \nabla \vec{U}^f) \right) = -\varphi \nabla p^f + \varphi \mu^f \nabla^2 [\vec{U}^f] + \varphi [\rho^f \vec{g}^f] + D
\]  
(2)

where \( \varphi \) is the soil porosity (-); \( \vec{U} \) is the velocity (m/s); \( \rho \) is the density (kg/m\(^3\)); \( t \) is time (s); \( p \) is pressure (Pa); \( \mu \) is the dynamic viscosity (Ns/m\(^2\)); \( g \) is the gravitational acceleration (m/s\(^2\)) and \( D \) an additional drag term (kg/(m\(^2\)s\(^2\))). The drag term was developed by Ergun [37] and accounts for momentum loss by means of fluid friction with the porous medium and flow recirculation within the sediment. To consider flow recirculation, an effective added mass coefficient is included after van Gent [38]. The porous drag term is defined as:
\[
D = -\left(150 \frac{1 - \varphi}{d_p} [\rho^f \vec{U}^f] + 1.75 [\rho^f \vec{U}^f] \nabla [\vec{U}^f] \right) 1 - \frac{1 - \varphi}{d_p} [\rho^f \vec{U}^f] \nabla [\vec{U}^f] - 0.34 \frac{1 - \varphi}{\varphi} [\rho^f \vec{U}^f] \frac{\partial [\vec{U}^f]}{\partial t}
\]  
(3)

with \( d_p \) (m) as effective grain size diameter.

PorousInter uses the volume of fluid (VOF) approach. Consequently, multiple phases are treated as one fluid with changing properties [39]. The indicator fraction \( \alpha (-) \) varies between zero for the air phase and one for the water phase. The water-air interface is captured by a convective transport equation:
\[
\varphi \frac{\partial [\alpha^f]}{\partial t} + \varphi \nabla \cdot ([\alpha^f] \vec{U}^f) = 0
\]  
(4)

The dynamic viscosity and the density of each fluid are calculated according to their fraction as:
\[
\mu = \alpha \mu_w + \mu_a (1 - \alpha)
\]  
(5)
\[
\rho = \alpha \rho_w + \rho_a (1 - \alpha)
\]  
(6)

The subscripts \( w \) and \( a \) denote the fluids water and air.

2.3. Turbulence

Turbulent properties have been captured by a large eddy simulation (LES) turbulence model (see also Section 3.1). Eddies up to a certain size were consequently directly resolved, whereas for small eddies a subgrid model is used. For the present study, the Smagorinski subgrid scale model [40] has been applied.

A measure \( M(\vec{x}, t) \) for the turbulence resolution was calculated after Pope [36]:
\[
M(\vec{x}, t) = \frac{k_t(\vec{x}, t)}{K(\vec{x}, t) + k_t(\vec{x}, t)}
\]  
(7)

where \( K(\vec{x}, t) \) defines the turbulent kinetic energy of the resolved motions by:
\[
K(\vec{x}, t) = \frac{1}{2} (\vec{U} - \vec{U}_{\text{mean}})(\vec{U} - \vec{U}_{\text{mean}})
\]  
(8)
and $k_r(x, t)$ defines the turbulent kinetic energy of the residual motions. The solver by Oxtoby et al. [34] and $k_r(x, t)$ defines the turbulent kinetic energy of the residual motions. The solver by Oxtoby et al. had to be adjusted to write $k_r(x, t)$ automatically. $K(x, t)$ and $k_r(x, t)$ were calculated and averaged for [34] had to be adjusted to write $k_r(x, t)$ automatically. $K(x, t)$ and $k_r(x, t)$ were calculated and averaged for

$K(x, t) = 0.25$ corresponds to a resolution of 25% of the turbulent averaged for the whole running time. A measure $K(x, t)$ corresponds to a resolution of 25% of the turbulent kinetic energy.

2.4. Boundary and Initial Conditions

Figure 2 shows the most important boundary conditions. The inlet of the boundary is divided into two fractions: for the air and the water phase. The parameter $a$ is fixed accordingly at the inlet. For the water phase, the elevation is set to $1$ m for case 1 and $0.25$ m for case 6. For the left, the water pressure condition is defined with a inlet pressure of $0$. The right boundary condition specifies that the total pressure for inflow and the dynamic pressure subtracted from the total pressure for inflow in Figure 2. For the weir, the impingement at the inlet is defined. The entrapment definition is applied up to the boundary and at the outlet. The stream is demanded by wall to be no slip. Consequently, no slip boundary conditions.

In OpenFOAM a definition of a constant water level at the outlet is challenging [41]. Therefore, a weir structure is established as a barrier to keep a constant water level for our model. The water flows freely over the weir top. Behind the weir, the water level decreases before it flows out of the model. This method is described e.g., in Bayon-Barrachina and Lopez-Jimenez [42]. For case 1, 5 and case 6 different height for the weir structure were chosen, since the water level is affected by the flow rate of the surface water. For the weir structure and at the whole bottom of the model, an impervious no slip condition is used. All boundary conditions in the third dimension contain slip conditions. The sediment is chosen as coarse sand with a grain size diameter of 2 mm and medium gravel with a median grain size diameter of 1.5 cm. The sediment is considered to be homogenous. The piping disturbance was modeled as a hydraulic head and force applied through the weir. These values are modeled by the solver, which is detailed in the next section.

2.5. Validation

To ensure reliable behavior of the integral model concerning the hydraulics for the interaction of groundwater and surface water, the solver was tested based on two applications. First, flow through a rectangular dam with a constant water level at both sides was investigated. The dam width amounts to $16$ m and the dam height to $24$ m. The dam height is equal to the water level at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam. A median grain size diameter of $2$ mm and a porosity of $0.25$ were defined at the left side of the dam.
The seepage calculated with the integral solver was in between the two-dimensional analytical solution after Di Nucci and the numerical solutions after Westbrook and Aitchison and Coulson. A large deviation was recognized for the one-dimensional solution. Based on the two-dimensional velocities observed in the numerical simulation, an analytical one-dimensional solution is obviously not adequate.

For the second validation case, the seepage through a homogeneous dam with a constant water level of 1.9 m on the left side was compared with an analytical solution by Kozeny [47] and an analytical solution by Casagrande [48]. The latter is an improvement of the solution by Kozeny. The dam has a height of 2.2 m and a width of 8.7 m. The two-dimensional mesh consists of 750,000 rectangular elements with a width of 0.02 m in the x- and y-direction. The dam material properties were the same as in the first validation case. A good agreement can be recognized for the simulated water levels with the calculated analytical data (Figure 4). At the entrance and at the outlet, the results gained with the integral model were closer to the solution after Casagrande compared to the solution after Kozeny.
Our test simulations showed that the integral flow model can predict the interaction of surface and groundwater with reasonable accuracy compared to analytical and numerical solutions.

3. Results and Discussion

In the following section, results for the reference case (see Table 1, case 1) will be presented for both the sandy and gravel sediment. Based on these results, the influence of the different ripple parameters and surface water discharge on the flow field will be analyzed, including pressure and velocity distributions as well as hyporheic exchange fluxes after 5 min simulation time. For all cases we focused on a single ripple in the center of a series of ripples. For the quantification of the fluxes, fluxes through the cell faces at the investigated ripple are calculated at the intersections of surface water and sediment. The ripple is divided into an area left and right from the ripple crest (see Figure 5). Inflow and outflowing fluxes for both sides as well as the sum of these fluxes divided by the face area—defined as “total flux”—are determined. The fluxes are averaged for the time frame of 60–300 s due to non-steady flow conditions.

3.1. Reference Case

For the reference case (see Table 1, case 1), the discharge amounts to 0.5 m$^3$/s, the ripple length to 20 cm and the height to 5.6 cm. Figure 6 shows the pressure distribution and velocity vectors at the investigated ripple (see Figure 1) for case 1 with a sandy and a gravel sediment. The solver solves the pressure term $p_{rgh}$ as the static pressure minus the hydrostatic pressure ($\rho g z$ with $z$ as coordinate vector). The highest pressure is observed at the last third of the upstream face of the ripple. Low pressure is present at the ripple crest and the first two-thirds of the upstream face as well as downstream of the crest. As these pressure differences lead to hyporheic exchange, flow occurs in downstream and upstream directions from high to low pressure. The described flow paths fit well to the results by Fox et al. [49], where the exchange of water between surface and subsurface was illustrated based on tracer experiments in the laboratory at a rippled sandy streambed. Also Thibodeaux and Boyle [50], Elliott and Brooks [14] and Janssen et al. [51] came to similar results from laboratory experiments with triangular bedforms. Fehlman [52] and Shen et al. [53] presented non-hydrostatic pressure distributions at triangular bed forms which were also similar to our results with pressure peaks at the middle of the stoss face, pressure minimum at the crest with low pressure remaining at the lee face until the pressure increases again at the stoss face of the following ripple. The description of the principal pressure pattern at the observed ripple in our simulations is valid for the sand as well as for the gravel, though the pressure values differ. Due to the higher resistance of the sand compared to gravel, higher pressure gradients are observed. Conversely, it behaves in terms of subsurface velocities: higher velocities are determined in the gravel sediment compared to the less permeable sand.

The applied LES turbulence model allows to resolve large parts of the turbulence at the streambed directly. Hence, between each ripple pair, eddies are identified. Comparing Figure 6 left and Figure 6 right, it is obvious, that the flow field in the surface water depends on the properties of the sediment: While in the sand, two eddies (clockwise as well as counterclockwise) can be recognized between the
were determined. For a better illustration of the non-Darcy-flow areas, Reynolds numbers up to 10 000 were recognized, while for the sandy bed Reynolds numbers up to 330 were claimed for gravel compared to sandy sediments.

The feedback from the subsurface to the surface is consequently also important. The clockwise eddies are located in the area where surface water enters into the ripple, while the counterclockwise ripple is located in the area where the water within the ripple flows back to the surface water.

### Table 2

<table>
<thead>
<tr>
<th>Case</th>
<th>Inflow Left (m³/s)</th>
<th>Inflow Right (m³/s)</th>
<th>Inflow Sum (m³/s)</th>
<th>Outflow Left (m³/s)</th>
<th>Outflow Right (m³/s)</th>
<th>Outflow Sum (m³/s)</th>
<th>Total Flux (m³/s/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.9 × 10⁻⁴</td>
<td>3.8 × 10⁻⁵</td>
<td>3.3 × 10⁻⁴</td>
<td>2.1 × 10⁻⁴</td>
<td>1.2 × 10⁻⁴</td>
<td>3.3 × 10⁻⁴</td>
<td>2.7 × 10⁻³</td>
</tr>
<tr>
<td>2</td>
<td>1.4 × 10⁻⁴</td>
<td>6.3 × 10⁻⁶</td>
<td>1.4 × 10⁻⁴</td>
<td>3.7 × 10⁻⁵</td>
<td>1.3 × 10⁻⁴</td>
<td>4.7 × 10⁻⁵</td>
<td>5.1 × 10⁻³</td>
</tr>
<tr>
<td>3</td>
<td>6.6 × 10⁻⁵</td>
<td>5.3 × 10⁻⁵</td>
<td>1.2 × 10⁻⁴</td>
<td>4.9 × 10⁻⁵</td>
<td>2.4 × 10⁻⁴</td>
<td>7.3 × 10⁻⁵</td>
<td>3.0 × 10⁻³</td>
</tr>
<tr>
<td>4</td>
<td>4.0 × 10⁻⁵</td>
<td>6.1 × 10⁻⁵</td>
<td>4.6 × 10⁻⁵</td>
<td>3.3 × 10⁻⁴</td>
<td>1.6 × 10⁻⁴</td>
<td>4.9 × 10⁻⁵</td>
<td>2.2 × 10⁻³</td>
</tr>
<tr>
<td>5</td>
<td>4.2 × 10⁻⁵</td>
<td>6.0 × 10⁻⁵</td>
<td>4.8 × 10⁻⁵</td>
<td>1.7 × 10⁻⁴</td>
<td>2.9 × 10⁻⁴</td>
<td>4.6 × 10⁻⁵</td>
<td>3.9 × 10⁻³</td>
</tr>
<tr>
<td>6</td>
<td>1.2 × 10⁻⁵</td>
<td>2.0 × 10⁻⁵</td>
<td>1.4 × 10⁻⁴</td>
<td>9.6 × 10⁻⁵</td>
<td>4.8 × 10⁻⁵</td>
<td>1.4 × 10⁻⁴</td>
<td>2.9 × 10⁻⁴</td>
</tr>
</tbody>
</table>

1 Total flux = (mag (inflow left) + mag (inflow right) + mag (outflow left) + mag (outflow right))/area.

Based on the overall high velocities within the sediment our simulations indicate, that non-Darcy-flow is present in the whole ripple nearly down to the bottom boundary for the gravel bed and to a part of the sandy bed (see Figure 7). At the near-surface area at the crest of the gravel ripple, Reynolds numbers up to 1770 were recognized, while for the sandy bed Reynolds numbers up to 330 were determined. For a better illustration of the non-Darcy-flow areas, Reynolds numbers up to 10 000 were required.
are illustrated in Figure 7. Consequently, dark red areas have a Reynolds number that equals or is higher than 10. Due to lower permeability, the flow velocities of the surface water influenced the sandy sediment less than the gravel bed with high permeability. The explicit modeling of the hyporheic zone with Darcy’s law is not possible in river beds with such coarse grain sizes since groundwater flow rates would be overestimated. Facing e.g., contaminant transport depending on residence time serious misperceptions could appear. The Reynolds number distribution of the following cases were similar to the reference case: for the whole gravel ripple down to the bottom non-Darcy-flow is apparent, while for the sand a small layer at the interface as well as the crest shows non-Darcy-flow areas. Only for case 5 with a distance of 20 cm between each ripple, there is even more non-Darcy-flow within the sandy ripple.

Table 3. Hyporheic fluxes of a single ripple in the center of a series of ripples for case 1–6 (gravel). Right and left indicate the part of the ripple right and left of the ripple crest (compare Figure 4).

<table>
<thead>
<tr>
<th>Case</th>
<th>Inflow Left (m³/s)</th>
<th>Inflow Right (m³/s)</th>
<th>Inflow Sum (m³/s)</th>
<th>Outflow Left (m³/s)</th>
<th>Outflow Right (m³/s)</th>
<th>Outflow Sum (m³/s)</th>
<th>Total Flux ¹ (m³/s/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.2 × 10⁻³</td>
<td>2.5 × 10⁻³</td>
<td>2.2 × 10⁻³</td>
<td>1.2 × 10⁻³</td>
<td>1.2 × 10⁻³</td>
<td>2.2 × 10⁻³</td>
<td>1.8 × 10⁻²</td>
</tr>
<tr>
<td>2</td>
<td>5.6 × 10⁻⁴</td>
<td>2.9 × 10⁻⁵</td>
<td>5.9 × 10⁻⁴</td>
<td>3.7 × 10⁻⁵</td>
<td>3.7 × 10⁻⁵</td>
<td>5.2 × 10⁻⁵</td>
<td>1.8 × 10⁻³</td>
</tr>
<tr>
<td>3</td>
<td>4.5 × 10⁻³</td>
<td>3.8 × 10⁻³</td>
<td>4.6 × 10⁻³</td>
<td>3.6 × 10⁻³</td>
<td>3.6 × 10⁻³</td>
<td>7.2 × 10⁻³</td>
<td>1.7 × 10⁻²</td>
</tr>
<tr>
<td>4</td>
<td>3.5 × 10⁻³</td>
<td>0</td>
<td>3.5 × 10⁻³</td>
<td>2.1 × 10⁻³</td>
<td>2.1 × 10⁻³</td>
<td>5.6 × 10⁻³</td>
<td>1.7 × 10⁻²</td>
</tr>
<tr>
<td>5</td>
<td>3.6 × 10⁻³</td>
<td>0</td>
<td>3.6 × 10⁻³</td>
<td>1.9 × 10⁻³</td>
<td>1.9 × 10⁻³</td>
<td>5.5 × 10⁻³</td>
<td>1.7 × 10⁻²</td>
</tr>
<tr>
<td>6</td>
<td>9.3 × 10⁻⁴</td>
<td>1.4 × 10⁻⁴</td>
<td>9.4 × 10⁻⁴</td>
<td>2.2 × 10⁻³</td>
<td>2.2 × 10⁻³</td>
<td>4.6 × 10⁻³</td>
<td>2.7 × 10⁻²</td>
</tr>
</tbody>
</table>

¹ Total flux = (mag (inflow left) + mag (inflow right) + mag (outflow left) + mag (outflow right))/area.

Figure 7. Reynolds numbers at a sandy (top) and gravel (bottom) ripple for case 1 (Table 1).

Janssen et al. [51] stated that the largest discrepancies of most CFD simulations of flow over ripples and dunes occur in the eddy zone. Especially for Reynolds-averaged Navier-Stokes turbulence models this is a known weakness. Therefore, we have chosen a LES turbulence model. At the same time, we are aware of the computational limitation, which is additionally increased by the
Janssen et al. [51] stated that the largest discrepancies of most CFD simulations of flow over ripples and dunes occur in the eddy zone. Especially for Reynolds-averaged Navier–Stokes turbulence models this is a known weakness. Therefore, we have chosen a LES turbulence model. At the same time, we are aware of the computational limitation, which is additionally increased by the calculation of the three-dimensional Navier–Stokes equations in the sediment in contrast to the commonly applied Darcy law. However, facing the growing availability of computational sources and the observed non-Darcy-flow areas in the investigated cases, we apply a promising tool for analyzing integral surface-subsurface flow processes with high resolution.

3.2. Ripple Dimension

For cases 2 and 3 the ripple length to height ratio is the same as for the reference case (see Table 1), but the ripple height and length are quartered for case 2 and doubled for case 3. Figure 8 shows the velocity and pressure distributions for the investigated ripples in the middle for case 2 sand and gravel. The general pressure pattern for case 2 for sand and gravel as well as for the reference case are similar: the lowest pressure occurs at the crest and the highest pressure upstream of the crest. But the high-pressure area related to the ripple size is much higher for case 2 than for the reference case. Related to the ripple face area at the interface, we consequently expect higher inflow rates compared to the reference case, which can be seen in Tables 2 and 3. The total flux per area is higher for case 2 with $5.1 \times 10^{-3} \text{ m}^3/\text{s}/\text{m}^2$ and $1.81 \times 10^{-2} \text{ m}^3/\text{s}/\text{m}^2$ than for the reference case with $2.7 \times 10^{-3} \text{ m}^3/\text{s}/\text{m}^2$ and $1.84 \times 10^{-2} \text{ m}^3/\text{s}/\text{m}^2$.
of one inflow area can be recognized at the upstream face of the ripple. Between these inflow areas, there is an outflow area. Another outflow area is located upstream of the lower inflow area, but the main outflow occurs downstream of the ripple crest. In the simulation of the gravel ripple, less eddies are observed than for the simulation with the sand. For the gravel ripple only one inflow area is present. The outflow is located similar to case 1 and 2: upstream from the inflow area and downstream from the crest.

The total fluxes per area are bigger for case 3 with sand ($3.0 \times 10^{-3} \text{ m}^3/\text{s/m}^2$) compared to the reference case (case 1) with sand ($1.8 \times 10^{-3} \text{ m}^3/\text{s/m}^2$). For the gravel the opposite is true (case 3, $1.7 \times 10^{-3} \text{ m}^3/\text{s/m}^2$ and case 1, $1.8 \times 10^{-3} \text{ m}^3/\text{s/m}^2$). The extremely high turbulence between the ripples for the sand could be an explanation for that. The results for case 2 and 3 with gravel and sand show, that a general statement about the influence of the ripple size is not possible, as there is a complex relation between the size and the material leading to different turbulence and pressure distributions, where also a threshold can be conceivable. Tonina and Buffington [16] presented results from a laboratory experiment with a pool-riffle channel and came to the same conclusion that hyporheic exchange does not necessarily decrease with lower bed form amplitudes. Closer investigations with more simulations including additional ripple size variations would be necessary for a more profound interpretation.
with more simulations including additional ripple size variations would be necessary for a more profound interpretation.

3.3. Ripple Length

For case 4 the ripple height equals the reference case, but the ripple length is doubled with 40 cm. This leads to higher pressure gradients for gravel and sand compared to the reference case. The pressure distribution is very similar to the reference case (case 1). The difference in pressure distribution leads to higher turbulence, which is higher for large height-to-length-ratios as already described by Broecker et al. [31].

3.4. Ripple Distance

Figure 11 shows the velocity and pressure distributions for case 5 with the same ripple geometry as for the reference case, but with a distance between the ripples of 20 cm. This distance leads to higher pressure gradients for gravel and sand compared to the reference case. The flow fields within the ripple area are similar to the reference case. But for this case there are also in- and outflow areas at the flat streambed between the ripples for both simulations. Eddies occur between the investigated ripples, but due to the distance, they are more elongated than for the reference case (case 1), since the pressure gradients are higher for case 5 compared to the reference case and the area is the same for both cases, (2.7 × 10^3 m^2). The total flux is higher for gravel as well as for sand compared to the reference case with both sediments (case 5: 3.9 × 10^3 m^2/s, case 1: 2.7 × 10^3 m^2/s). Broecker et al. [31] already presumed higher hyporheic exchange for this case compared to the reference case based on the higher pressure gradients. For the knowledge of distances between the ripples, the authors investigated surface water processes, apart from Broecker et al. [31] where only a surface water model was used.
Compared to the investigated sandy ripples described above, the non-Darcy flow areas of case 5 are significantly larger (compare Figures 5 and 12). Due to the distance between the ripples, higher flow velocities reach the ripple stoss which influence the velocities within the ripple.

Figure 11. Pressure distribution and velocity vectors at a sandy (left) and gravel (right) ripple for case 5 (Table 1). The color indicates the pressure distribution and the arrows indicate the velocities. The scaling is different in the right and the left panel. The arrows indicate the direction of the subsurface flow. The intensity of the subsurface flow is used.

For case 6 the discharge was set to 0.25 m$^3$/s (for case 1–5 the discharge was 0.5 m$^3$/s). The ripple geometry is the same as for the reference case [case 1]. Comparing the reference case with case 6 it is obvious that both flow discharges show qualitatively similar flow fields. The flow velocities within the ripples decrease due to lower surface water velocities. Nevertheless, there is still a layer with Reynolds numbers higher than 50, which is similar to that of the reference case. Reynolds numbers higher than 50 indicate the transition to the flow regime of the beach ripple case (see Figure 5).

Figure 12. Reynolds numbers at a sandy ripple for case 5.
4. Conclusions

CFD simulations were designed to simultaneously examine both surface and subsurface flow processes with an extended version of the three-dimensional Navier–Stokes equations. Based on two simulations for seepages through dams, it is shown that the applied model can describe the interaction of groundwater with surface water. The coupled CFD model was applied to investigate the impact of bed form structures on flow rates and surface form changes on large areas in the hyporheic exchange.

The examined ripple structures changed the streambed pressure and created in- and outflowing fluxes within the sediment. For the sandy sediment, the non-Darcy-flow areas are restricted to the upper layers of the sediment, whereas in the gravel sediment higher pressures were observed in the subsurface areas. The interface is highly turbulent due to increased pressure areas, surface water flows into the ripple, while subsurface water flows out of the ripple towards the stream.

Comparing the extended Navier–Stokes equations with the commonly used coupling of surface water with a Darcy-law-model, the integral model is definitely more time consuming than the coupled models. The model shows direct feedbacks from surface to subsurface and vice versa, which is not commonly achieved in many models. The model is applicable also in non-Darcy-flow areas and provides a high resolution in results, which can be applied in further studies.

The general flow paths were the same for all simulations. Upstream of the crest in high pressure areas, surface water flows into and out of the ripple, while the one below is upstream of the riverbed. Downstream, the flow is generally directed away from the riverbed. The effective viscosity has to be determined.

Numerous of the observations of our simulations were already seen in laboratory experiments. The model shows direct feedbacks from surface to subsurface and vice versa, which is not commonly achieved in many models. The model is applicable also in non-Darcy-flow areas and provides a high resolution in results, which can be applied in further studies.
the interface and to determine in- and outflowing fluxes, which can be important for the understanding and prediction of hydrological, chemical, and biological processes. In contrast to other coupled models, it is applicable in non-Darcy-flow areas and allows to simultaneously simulate the surface and subsurface with one system of equation for surface and groundwater. We can develop upscaling approaches where we quantify the exchange rates depending on the ripple geometry and other variables with the high resolution three-dimensional integral model to serve as sink/source terms in one- or two-dimensional shallow water flow models. The shallow water equations are based on vertical averaged velocities (not discretizing the vertical dimension) and are generally applied on coarser scales. In a next step, also transport equations will be included in the presented integral model.

Author Contributions: Methodology, T.B. and K.T.; Software, T.B. and K.T.; Validation, T.B.; Investigation, T.B. and V.S.G.; Writing—original draft, T.B.; Writing—review and editing, T.B., R.H., G.N. and J.L.; Visualization, T.B.; Supervision, R.H. G.N. and J.L.

Funding: The funding provided by the German Research Foundation (DFG) within the Research Training Group ‘Urban Water Interfaces’ (GRK 2032) is gratefully acknowledged.

Acknowledgments: Parts of the simulations were computed on the supercomputers of Norddeutscher Verbund für Hoch- und Höchstleistungsrechnen in Berlin.

Conflicts of Interest: The authors declare no conflict of interest.

References
2. Brunke, M.; Gonser, T. The ecological significance of exchange processes between rivers and groundwater. *Freshw. Biol.* 1997, 37, 1–33. [CrossRef]


42. Bayon-Barrachina, A.; Lopez-Jimenez, P.A. Numerical analysis of hydraulic jumps using OpenFOAM. *J. Hydroinform.* 2015, 17, 662–678. [CrossRef]

© 2019 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/).