Spiller, Stephan Mark; Rüther, Nils; Belete, Kiflom; Strellis, Brandon

Assessing environmental effects of hydropower peaking by 3D numerical modeling

Assessing environmental effects of hydropower peaking by 3D numerical modeling

Stephan Mark Spiller, Nils Rüther, Kiflom Belete, Brandon Strellis

The aim of the present study is to evaluate whether 3D numerical modeling is suitable to predict frequent load fluctuations in natural rivers according to different hydro power operation schemes. The study is carried out in the framework of the Center of Environmental Design of Renewable Energies (CEDREN). The objective of the centre is to develop and disseminate effective design solutions for renewable energy production that take adequate account of environmental and societal issues, both locally and globally. The focus of the present study is on the simulation of highly unsteady flow, including strongly varying water levels in a natural river reach of approximately 1.2 km length and 75 m width. The flow in the river reach is highly complex, since it includes river bends as well as large pool / riffle sequences. The model will be verified by measurements of 3D velocities and water levels. In the future, these results will be used to investigate morphodynamic changes and risk for potential fish stranding areas.

Das Ziel der vorliegenden Studie ist es ein dreidimensionales, numerisches Programm auf einen Flussabschnitt mit lokal und zeitlich starken Wasserspiegellagenschwankungen anzuwenden und zu beurteilen ob dieses numerische Programm in der Lage ist die richtigen Wasserspeiegellagen vorherzusagen. Die Studie ist im Rahmen des Center of Environmental Design of Renewable Energy (CEDREN) durchgeführt. Das Ziel des Forschungsprojektes ist die Entwicklung und Verbreitung wirksamer Lösungen für erneuerbare Energien, welche ökologische und gesellschaftliche Problematiken in Betracht ziehen. Der Fokus der vorliegenden Studie beruht auf der Simulation von stark instationären Abflüssen einschließlich stark schwankender Wasserstände in einem natürlichen Flussabschnitt von ca. 1.2 km Länge und 75 m Breite. Die Strömung in diesem Flussabschnitt ist aufgrund von Flussschleifen und ausgeprägten Pool / riffle Sequenzen sehr komplex. Die Simulationsergebnisse werden mit gemessenen Wasserständen verglichen. In Zukunft werden die Resultate dieser Studie genutzt um Untersuchungen in den Bereichen morphodynamischer Veränderungen sowie Risiken möglicher Fisch Strandung durchzuführen.

1 Introduction

In the near future the production of electricity through hydropower peaking will increase dramatically. Especially Norway, where geographical situation and easy availability of the natural resource water favor hydropower peaking, will experience changes in electric power production schemes. When meeting peak load requirements, a power station is turned on at a particular time during the day. It generates power at a constant load for a certain number of hours and is then turned off or set to a different load for another time period, resulting in a high variability of flow discharges. Where reservoir hydro schemes are operated primarily to provide peak load services, particular environmental risks should be considered in any environmental impact assessment. At a minimum, focus should be on water quality, fluvial geomorphology, riparian vegetation, macro invertebrate, and fish communities underpinned by a sound hydrological analysis.

The most important issues are fast and frequent water level changes when stopping or starting production. From a morphological point of view this may result in an increased seepage-induced erosion of riverbanks as well as in special sorting processes of bed substrate, resulting in reduced hydraulic conductivity as well as clogging processes due to sedimentation of fines. From an ecological point of view this may have an impact on macro invertebrate and fish communities exposing them to flash floods and an increased stranding risk.

The motivation for the study was therefore to apply a 3D numerical code to a natural river reach in order to evaluate whether the program is usable to predict harm to the environment. Using CFD in hydraulic engineering applied to natural rivers has a young history. Rüther et al (2010) gives a short overview of CFD models applied to natural streams. Baranyi & Józsa (2006) modeled the flow characteristics of a 4 km long reach of the river Danube with a CFD code called SSIIM (Olsen 2010) and concluded that measured velocities and turbulent kinetic energy are well reproduced by the model. Abad et al (2008) modeled the complex flow in a bend with 5 subsequent, submerged weirs over the total length of approximately 200 m and 20 m width with the commercial CFD code Flow3D 9.0 (Flow Science, Inc., Santa Fe, N.M.). They concluded that the CFD model reproduced the general pattern of the flow through the reach, including the recirculation zones in the vicinity of the weirs. All the before mentioned studies used the Reynolds-averaged Navier-Stokes equations and the standard k-Epsilon turbulence closure. The simulations were carried out for a steady state discharge. In all the cases, the flow is not subject of a flow change from subcritical to super critical.

Therefore, the present study aims to use a commercial CFD code to simulate highly unsteady flow with strongly varying discharge and consequent strongly varying water levels in a river reach with alternating pool and rapid sequences. The commercial code is called Star CCM+ (CD-adapco, Melville, NY, USA) and is introduced in chapter 3. The experimental field site is introduced in chapter 2. Chapter 4 describes the results and chapter 5 sums up the main conclusions and gives an outlook to further research.

2 Field Site

The River Nidelva has its catchment in mid Norway, south-east of Trondheim. The Nidelva watercourse has its source near Sylan in Sweden and flows west wards out to sea in Trondheim. The distance from the source to the sea is 160 km and the catchment area of the watercourse is 3100 km^2. The river Nidelva is characterized by several hydro power plants and is therefore highly regulated. The investigated study reach, shown in figure 1, is approximately 1200 m long and is located 5 km south of Trondheim. Located at the upstream boundary of the study reach, is the outlet of the last hydropower plant before river Nidelva flows into the Trondheim fjord. Downstream of the power plant, the river is characterized by sequences of large bended pools and rapids as it is depicted in figure 1.

Right after the upstream pool, the water heads for a rapid and changes from sub to supercritical flow. As it can be seen in the aerial photo in figure 1, this section is characterized by strong surface waves and local hydraulic jumps. The rapid continues throughout the subsequent bend and leads to a pool which ends in the models downstream boundary condition. The bed elevation difference between the end of the upstream pool and the beginning of the downstream one, is approximately 3.0 m

The data to establish the digital elevation model and the grid for the numerical simulation was gained form digital maps with detailed elevation lines as well as bathymetry measurements. The bathymetry measurements were taken by boat, kayak and wading. The measurements taken by boat and kayak are based on a single beam echo sounder measurement coupled with a differential global positioning system (D-GPS). The measurements taken by wading are based on D-GPS only. An upstream cross section was subject of Acoustic Doppler Current Profiler (ADCP) measurement.

Figure 1 Aerial photo of the study site. Flow direction from down right to up left.

3 Numerical method

The commercial CFD software Star-CCM+ (CD-adapco 2010) uses a volume-of-fluid (VOF) method to solve various engineering problems. The application deals with Certificate Trust List (.stl) files to remesh the geometry surface for further handling. To create a volume mesh, it offers three techniques to choose from. Those are tetrahedral, trimmed (hexahedral) and polyhedral cell shape based core mesh models. For more accurate observation of important regions, a finer grid can automatically be generated around derived parts.

Star-CCM+ provides several physical models for an efficient simulation, including Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) turbulence models.

For the current study, a trimmed (hexahedral) mesh was generated with an anisotropic cell size distribution in the z-direction. This leads to comparatively shallow cells along a slightly inclined plane in the position of the water surface level. The cell height increases with the distance to the water surface.

The base size for the x and y extension of the cells is 3m while the height of the cells varies from 10% of the base size (0.3m) in the area around the water surface level, to several meters in regions with less required accuracy. The total number of cells is 7.600.000.

For the present simulation, describing a multiphase flow of water and air, a three dimensional, implicit unsteady, gravity driven, segregated flow model together with a k-Epsilon, RANS turbulence model was chosen.

4 Results

Figure 2 displays the digital elevation model, used for the simulation. The observed region spans altitudes between 0.25 m and 66.25 m over the mean sea level, while several pool structures are visible along the course of the river.

The alternation of pool and riffle sequences produces clear rapid areas that could very well be reflected in the numerical results. Figures 3 and 4 show the water surface of the steady state flow before and after a hydropower peaking operation, shaded by the surface velocity magnitude. While figure 3 gives an overview over the entire study reach, figure 4 offers a closer look upon the rapid sequence in the center of the study reach, from a perspective point of view.

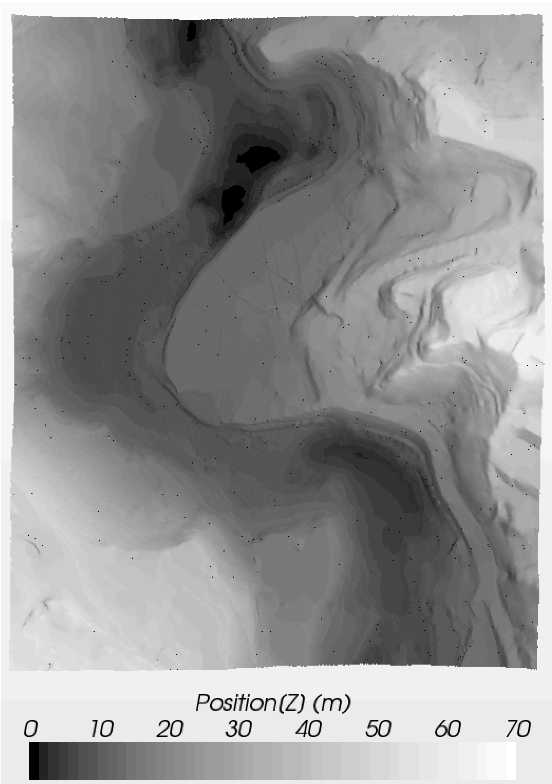

Figure 2 Digital elevation model of the study reach

The simulated scenario begins with the development of a steady state river flow with a discharge of 150 m^3/s (see figures 3 and 4 left). During this state, the water depth at the deepest point of the upstream pool was about 8.5 m with a surface velocity of 0.6 m/s. The following section of rapids accelerated the mean flow up to 4 m/s. Some scattered points within the rapids, even reached velocities just over 5 m/s. Short before the outlet boundary, another pool slowed the flow down to a mean velocity of about 1 m/s. Recirculation zones, developed along the undercut slopes of the river and appear as dark areas in figure 3 and 4.

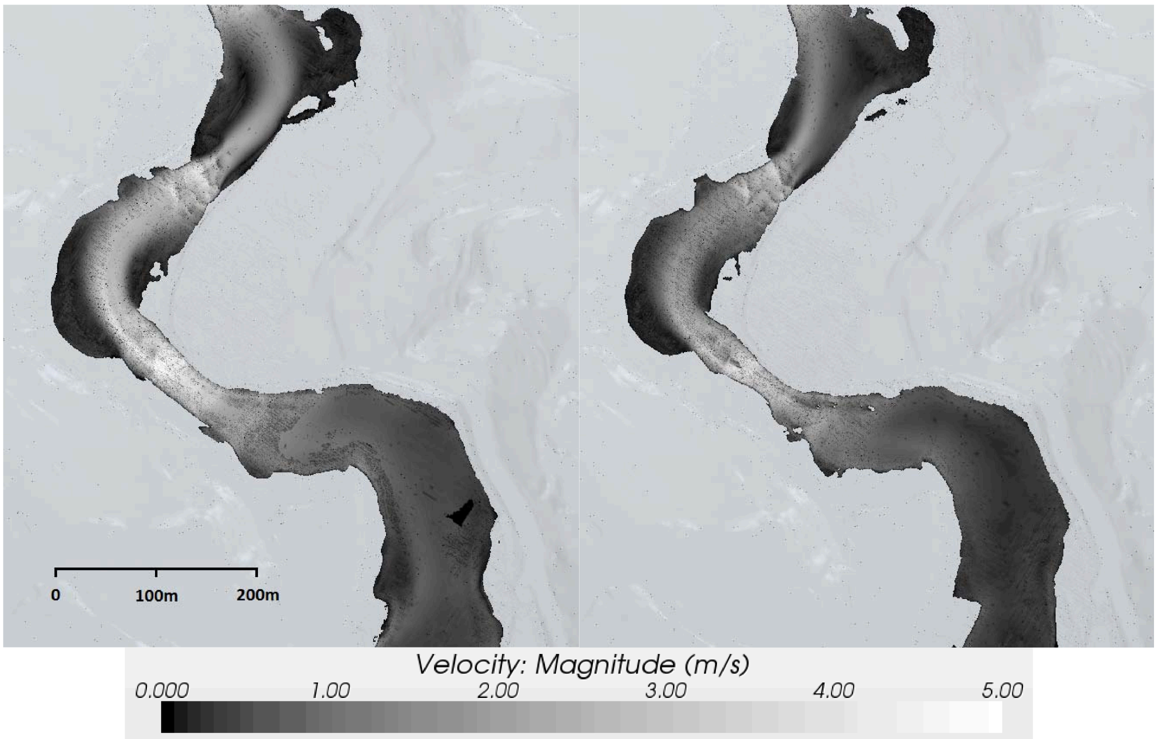

Figure 3 Water surface location and velocity at 150m^3/s (left) and 50m^3/s (right)

After the achievement of this fully developed situation, a hydropower peaking operation was initiated. During 20 minutes, the inflow discharge was decreased linearly from 150 m^3/s to 50 m^3/s. This extreme event corresponds to a negative ramping rate of 5$m^3/s/min$. Figures 3 and 4 (right), show the water surface of the fully developed flow after the hydropower peaking scenario. Due to less discharge, the mean water level was lowered. In the upstream pool, a new maximum water depth of 8 m was reached while the surface velocity in this part was 0.3 m/s. Despite the lower discharge, the velocity along the rapid sequences remained comparatively high, with magnitudes of about 3 m/s. This corresponds to the much slighter width of the river in this part, compared to the situation before the hydropower peaking event. Besides the velocities, also the wetted area within the riverbed was significantly reduced. Parts of the river dried out and left several areas for potential fish stranding. Furthermore, the expanse of the recirculation zones was reduced as a consequence of the lower discharge and water level.

Throughout the highly unsteady process of the hydropower peaking scenario, a constant dry out of the riverbed began to develop. With its origin at the inlet of the study reach, over time this effect was carried forward towards the downstream boundary. The discharge at the outlet, located approximately ca. 1200 m away from the inlet, showed a first response 8 minutes after the beginning of the inflow decrease. Almost steady state conditions were reached about 45 minutes after the development of the new inflow discharge.

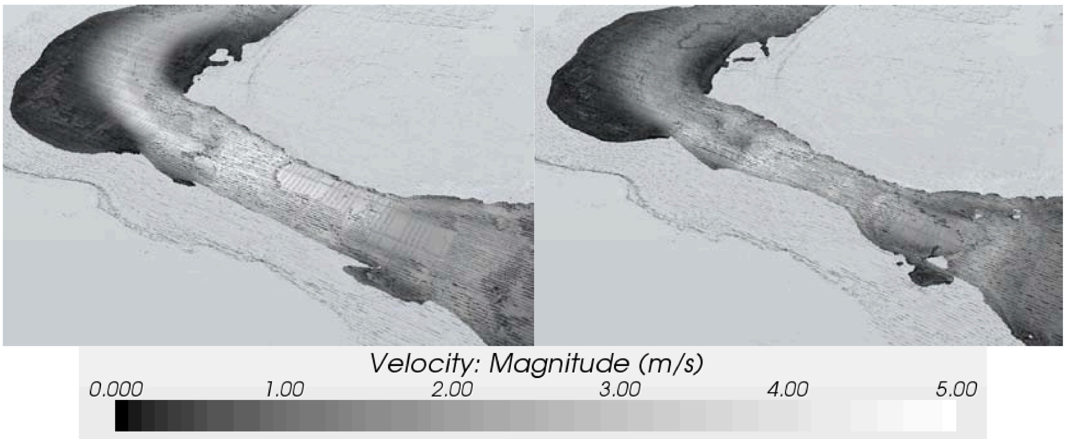

Figure 4 Water surface velocity in rapid area at 150 m^3/s (left) and 50 m^3/s (right). Perspective view.

5 Conclusion and further research

The present study documents the flow simulation of a highly unsteady flow process with a commercial 3D CFD code. The modeled natural river reach is 1200 m long, 75 m wide and is characterized by alternating pool and rapid sequences. Throughout the simulated scenario, the discharge is reduced from 150 m^3/s to 50 m^3/s with an extreme negative ramping rate of 5 m^3/s/min.

The study shows that it requires a large amount of CPU power to simulate such a complex flow at highly unsteady conditions. However, the study shows also that the results of the simulation have great potential for the investigation of wetted areas as well as shear stress and velocity distribution along the reach during a strong decrease of discharge. The authors see a great potential in the fact that the model was able to calculate the flow change from areas with sub critical flow, to flow in rapids dominated by high velocities and severe surface waves.

The goal of this study was to identify a CFD model which is capable of simulating flow changes in natural rivers with highly unsteady flow rates. Further studies will use an improved digital elevation model along the rapids and will use measurements of averaged cross sectional velocities and shear stress estimations to verify the numerical model. In addition, the model will be applied to cases of extreme discharge increase in order to study additional forces on the river bed. Finally, the model range will be extended to areas further downstream where existing data of stranding areas is available.

References

Abad, J. D. Rhoads, B. L. et al.: Flow structure of different stages in a meander-bend with bendway weirs. In: Journal of Hydraulic Engineering 134(8), (2008), 1052-1063.

Baranya, S. Józsa, J.: Flow analysis in river Danube by field measurement and 3D CFD turbulence modelling. In: Periodica Polytechnica: Civil Engineering 50(1), (2006), 57-68.

CD-adapco: User Guide Star-CCM+ Version 5.06.007, (2010).

Rüther, N. Jacobsen, J. Olsen, N. R. B. Vatne, G.: Prediction of the three-dimensional flow field and bed shear stresses in a regulated river in mid-Norway. In: Hydrology Research 41(2), (2010), 145-152.

Olsen, N.R.B.: SSIIM manual, http://folk.ntnu.no/nilsol/ssiim/manual3.pdf (2010)

Authors:

Stephan Mark Spiller, M.Sc.,
Dr. Nils Rüther (PhD),
Dr.-Ing Kiflom Belete
NTNU, Norway,
S.P. Andersens veg 5,
N-7491 Trondheim,
Tel.: +47 - 735 92409
stephan.spiller@ntnu.no

Brandon Strellis
Mason Building
790 Atlantic Drive
Atlanta,
GA 30332-0355 P