Conference Paper, Published Version

Rüther, Nils; Pedersen, Øyvind

3D numerical modeling of the flow over a gravel river bed due to hydropower peaking

Dresdner Wasserbauliche Mitteilungen

Vorgeschlagene Zitierweise/Suggested citation:
Rüther, Nils; Pedersen, Øyvind (2014): 3D numerical modeling of the flow over a gravel river bed due to hydropower peaking. In: Technische Universität Dresden, Institut für Wasserbau und technische Hydromechanik (Hg.): Simulationsverfahren und Modelle für Wasserbau und Wasserwirtschaft. Dresdner Wasserbauliche Mitteilungen 50. Dresden: Technische Universität Dresden, Institut für Wasserbau und technische Hydromechanik. S. 375-382.

Technische Universität Dresden – Fakultät Bauingenieurwesen
Institut für Wasserbau und Technische Hydromechanik

37. Dresdner Wasserbaukolloquium 2014
„Simulationsverfahren und Modelle für Wasserbau und Wasserwirtschaft“

3D numerical modeling of the flow over a gravel river bed due to hydropower peaking

Nils Rüther
Øyvind Pedersen

Hydropower peaking has become extremely popular among the energy suppliers when it comes to production of energy from an electrical hydropower plant. Electricity is produced in times of the day where the demand is high. Since these periods are short and frequently reoccurring, it can happen that the water level in the river is changing heavily during one day. The effects of this frequent changes has been studied recently, however many effects remain yet unknown, e.g. the effects of the frequent flow changes on the river bed. There are several hypotheses on how the river bed is behaving. However, there are few data sets available that describes the total distribution of bed shear stress over time in a river. Therefore does this study attempt to predict the wetted area for an unsteady flow event by the means of computational fluid dynamics (CFD). A commercial CFD program is used to predict the flow velocities and the wetted area during a hydro power peaking event. The program works with a finite volume technique to solve the Reynolds-averaged Navier Stokes equations which are closed by the standard k-ε turbulence model. The goal of the study was to investigate the capability of the models to simulate hydrodynamic effects from the unsteady flow on the river bed. Measurements of discharge, waterlines and flow velocities were taken at one flow rates. The results showed that the program is able to calculate the ramping rate given very well. The flow pattern, the velocity distribution and the calculated water covered areas matched well.

Stichworte: CEDREN, hydro power peaking, CFD, RANS, gravel bed, armoring

Block 5
Saal 3

1 Introduction

In the near future the production of electricity through hydropower peaking will increase dramatically. Especially Norway will experience changes in electric power production schemes. Geographical situation and easy availability of the natural resource water favor hydropower peaking. 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. At power stations which are run with hydro power peaking, environmental harm should be addressed in any environmental impact assessment. Focus should be on water quality, fluvial geomorphology, riparian vegetation, macro invertebrate, and fish communities underpinned by a sound hydrological analysis.

To investigate these effects of hydropower peaking on the river and its environment, the use of numerical has become extremely popular. Especially 3D CFD codes have been used successfully to simulate the flow field in natural rivers to make prognosis on water levels or bed shear stress. *Rüther et al.* (2010) and *Spiller et al.* (2011) or *Olsen & Haun* (2010) give a good overview over available models and their application.

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. *Spiller et al.* (2011) modeled a decreasing discharge in a natural river reach to identify potential stranding risk areas.

Being able to realistically model the flow conditions during a hydro-peaking event is an important part of assessing the environmental risks. This study aims to assess the possibility to use the commercial code Star CCM+ to simulate highly unsteady flow with strongly varying discharge and consequent strongly varying water levels in a bended river reach where the bed is characterized by a strongly developed armored gravel bed layer.

2 Field site and data base

The investigated river reach is a part of river Lundesokna which is located in the center of Norway, ca. 30 km south of Trondheim and can be seen in Figure 1. The top view shows the 200m long and 20 wide river reach. The flow is from left to right. The outlet of the hydro power tunnel is roughly 1 km upstream.

In order to create a digital elevation model as input for the numerical model, the bathymetry was measured. At low water levels as laser scanner was used to accurately measure the elevation of each measured point. The resulting point cloud had a point spacing of 4-40 cm. The bathymetry of the water covered area was measured with differential GPS with a spacing of 0.5 – 1.5 meters.

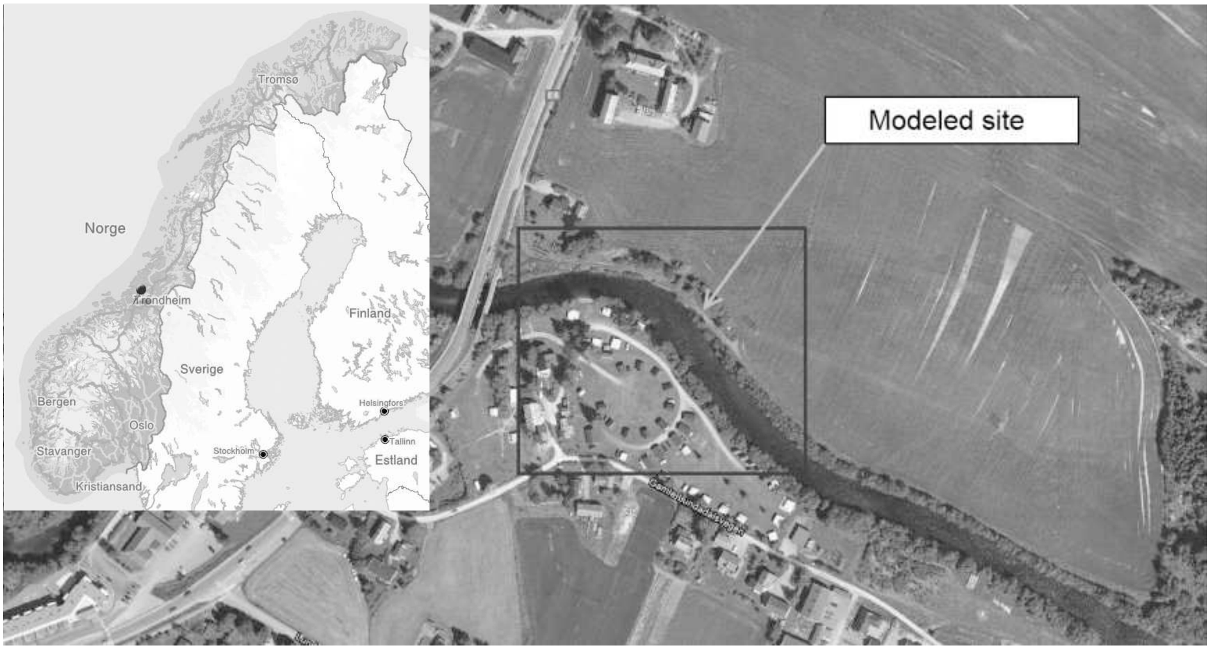


Figure 1: Airborne photo the investigated reach; the flow direction is from right to left

At one discharges the velocity over the width were measured with an ADCP M9 of Sontek. In addition to that, the detailed water line was measured with the differential GPS. This is to compare the calculated and the measured water covered area. Water covered area is a very important parameter to asses when talking about the environmental impact of hydro power.

The river reach is heavily regulated and leads at its end to Gaula river, one of Norways best salmon rivers. The substrate at the river bed is strongly armoured and clogged. The surface roughness was determined by characteristic grain sizes of two freeze cores which very exemplarily taken in the middle of the river and on the side.

3 Numerical method

The commercial CFD software Star-CCM+ (*CD-Adapco*, 2011) 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 turbulence closure models for an efficient simulation, including Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) turbulence models.

The Volume of fluid method (VOF) is commonly used for two-phase flow in commercial programs. It defines a volume fraction of each phase in a cell. It is assumed that the velocities and pressures are the same for both phases within the cell. A transport equation is used to calculate the change in volume fraction over a time step, and a filtering method can then be used to find the location of the free water surface.

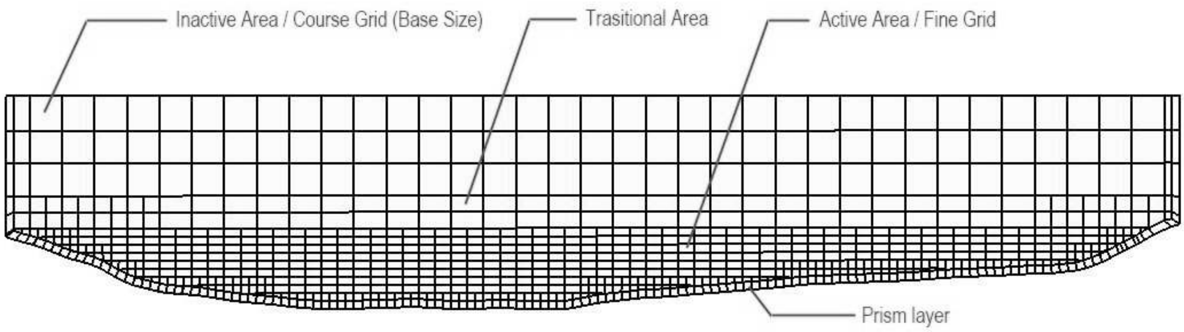


Figure 2: Cross section with the used grid displayed

For the current study, a grid with trimmed cell shapes was used. A finer grid was used to model the active, water filled area. The total number of cells was about 3,000,000. The grid used is a non-orthogonal, unstructured grid with fixed cells. Star-CCM+ also provides the possibility to make extrusion meshes, extending the conditions at the boundary (Figure 2).

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, realizable RANS turbulence model was chosen. Star CCM+ uses a y+ wall treatment with varying modeling assumptions for fine or course grids to model flow close to the walls. There exist several numerical discretization techniques for solving the convection-diffusion equation in CFD. This model uses a second order upwind scheme, which typically spends longer time to converge than a first order scheme, but is more accurate.

4 Results

4.1 Steady state simulations

To verify the simulation results, the calculated velocities in one cross section at a discharge Q = 20.63 m3/sec have been compared with ADCP measurements In addition the plan view location of the water edge for two different discharges have been compared.

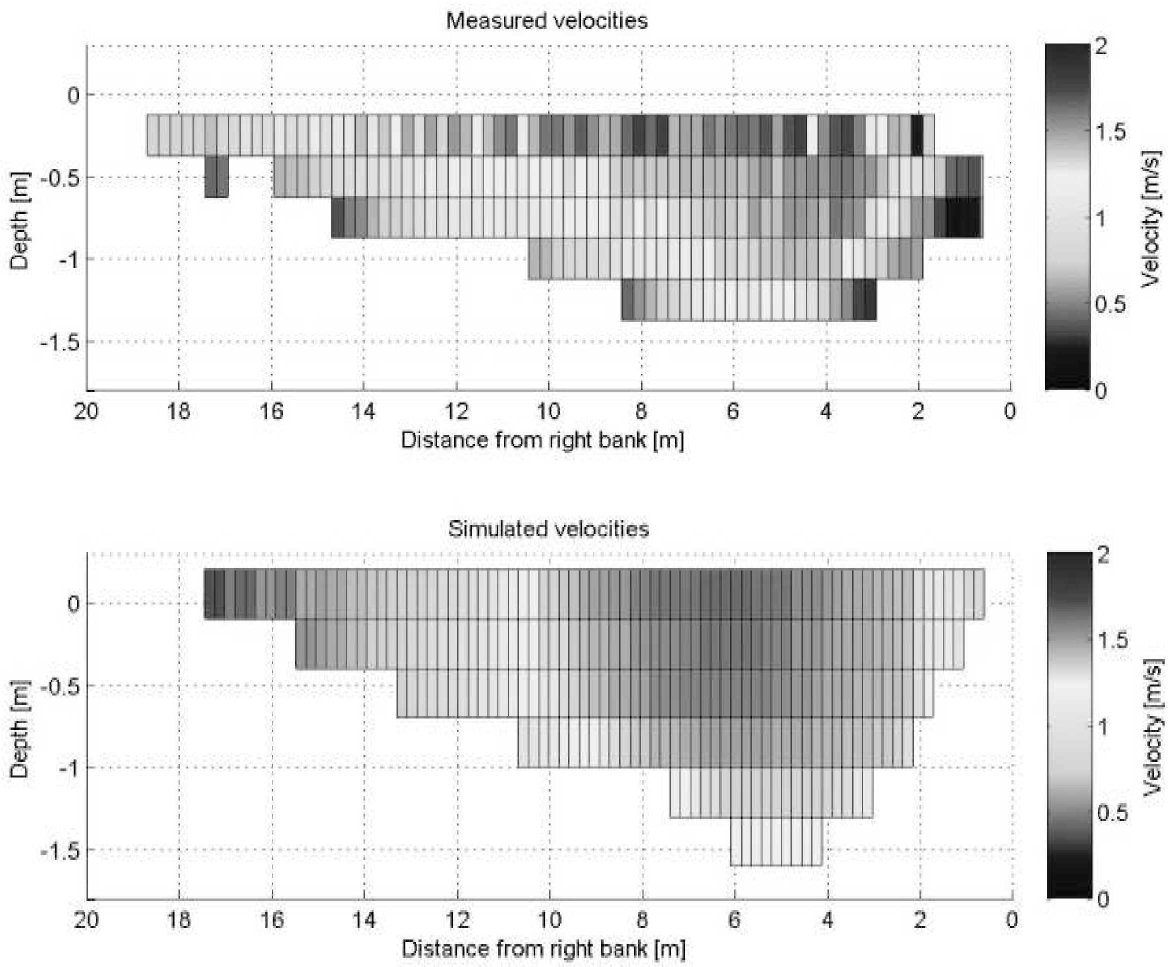
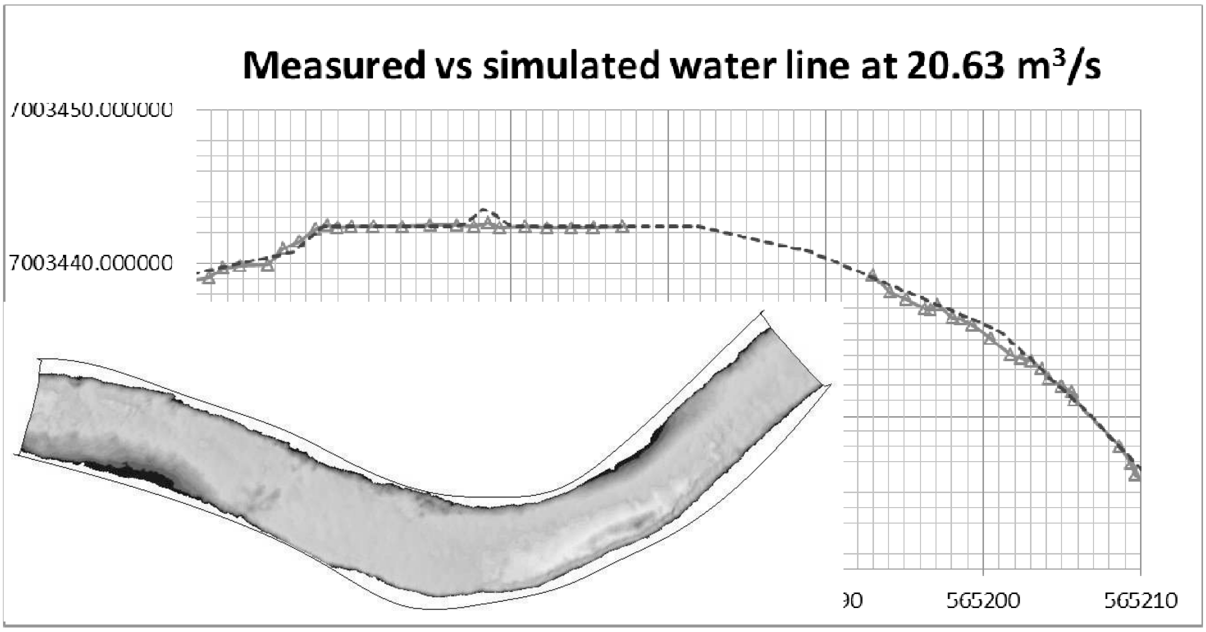


Figure 3: Above the measured, below, the simulated velocities at Q = 20.63 m^3/s.



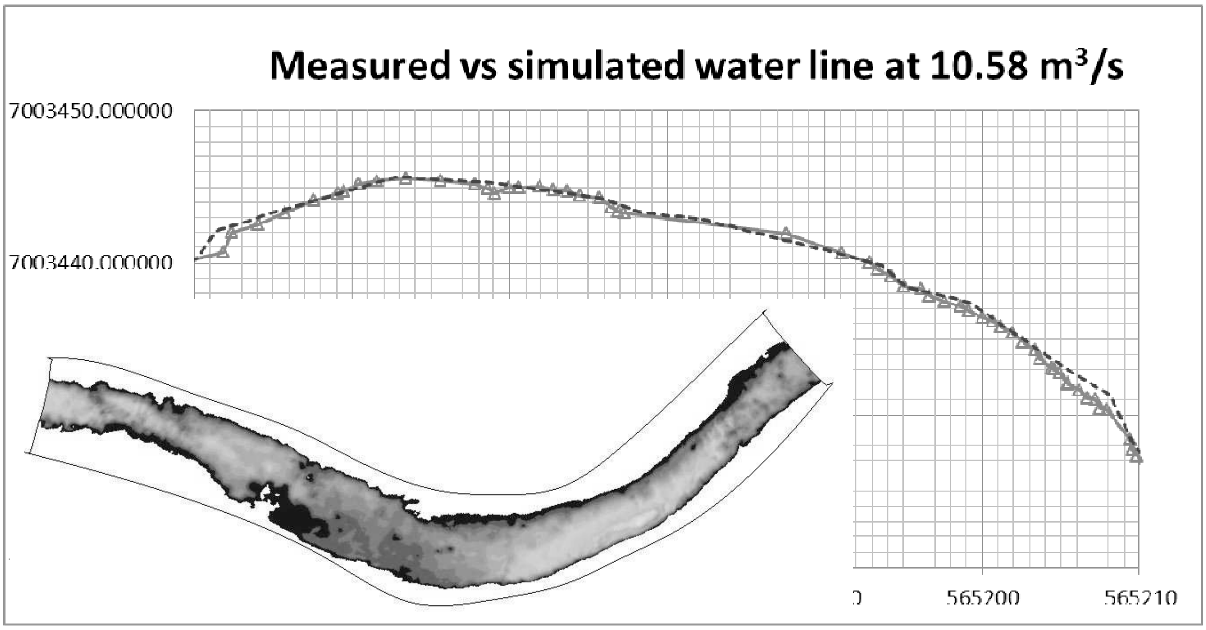


Figure 4: Water covered area for 2 different discharges.

Figure 3 shows the measured and the calculated velocities in one cross section. One can see that the comparison matches well. The maximum velocities of about 2 m/s are well calculated. Also the location of the center area with the maximum velocities matches well if one compares the measured and simulated ones. In the measurement, this location of the maximum velocity seems to be slightly shifted towards the outside of the bend.

Figure 4 shows the comparison of the water covered area for two discharges. At the top of the figure, Q = 20.63 m^3/s and at the bottom Q = 10.58 m^3/s. Also here, the results of the calculation match very well with the measurements. The measured water line from the left side of the river is almost identical for both discharges. The plan view figures in Figure 4 show the surface velocity, where high velocities are illustrated in yellow/ red and lower ones in blue.

4.2 Unsteady state simulations

After the verification of the flow in the steady state simulation the authors wanted to test the possibilities to model a highly unsteady case. These results were to compare against measurements taken during a peaking event resulting of a turbine start at the power plant. The discharge Q increased from 2 to 20 m^3/sec in 2 min. This resulted in a ramping rate of 9 m^3/s per minute. The results are illustrated in a movie and compared visually to the video recording of the real situation.

Clearly the most rapid changes can be observed within the first minute of the experiment. This is due to the shape of the river at that cross section. After that a recirculation at the inner side of the bend and this can also be seen in the simulation. However at this stage of the video there is a wave effect in the simulation which does not seem very physical. That means the wetting of the shallow area of the inner gravel bank is overestimated by the model. In reality the bank is wetted rather in a lateral direction. One can also see the lower velocities at the boundary and that the water line increases in roughly the same speed. Overall one can say that the progression speed of the wave and the increasing water covered area during the peaking event matched very well.

5 Conclusion and further research

The present study documents the flow simulation of a highly unsteady flow process with the commercial 3D CFD code Star CCM + developed by CD-Adapco. The modeled natural river reach is 200 m long, 20 m wide and is characterized by an alternating pool and rapid sequence. First several steady state discharges are simulated and verified with the measurements. To test the codes ability to handle highly unsteady, increasing, flow, a peaking event due to a turbine start is simulated. Throughout the experiment, the discharge is increased from 2 m^3/s to 20 m^3/s with appositive ramping rate of 9 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 increase 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 measurements of averaged cross sectional velocities and shear stress estimations during the peaking event to verify the numerical model.

Acknowledgements:

The study was financed by CEDREN (www.cedren.no). The authors are very thankful to CD-adapco, who was willing to share the CFD code Star CCM+ with academia.

6 Literatur

CD-Adapco. (2011). User Guide, Star CCM+ Version 6.06.011. CD-Adapco.

Olsen, N., & Haun, S. (2010). Free surface algorithms for 3D numerical modelling of reservoir flushing. RiwerFlow 2010. Braunschweig, Germany.

Rüther, N., Jakobsen, J., Olsen, N., & Vatne, G. (2010). Prediction of three dimensional flow field and bed shear stress in a regulated river in Mid-Norway. Hydrology research, 41.2, 145-152.

Spiller, S., Rüther, N., Belete, K., & Strellis, B. (2011). Assessing environmental effects of hydropower peaking by 3D numerical modeling. Annual conference on hydraulic engineering, 34th proc. Dresden: Technical University.

Autoren:

Dr. Nils Rüther

Department of Hydraulic and Environmental Engineering
Norwegian University of Science and Technology, Trondheim, Norway
S.P. Andersensveg 5
N-7491 Trondheim

Tel.: +47 951 377 47
Fax: +47 359 129 8
E-Mail: nils.ruther@ntnu.no

Øyvind Pedersen

Multiconsult AS
Nedre Skøyen vei 2
0276 Oslo