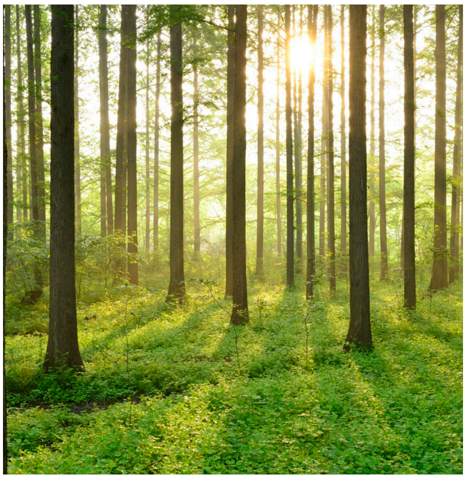
1D SYSTEM TRANSIENTS COUPLED WITH 3D LOCAL FLOW DETAILS

REPORT 2015:109



HYDROPOWER

Energiforsk

1D system transients coupled with 3D local flow details

OLIVIER PETIT

ISBN 978-91-7673-109-3 | © 2015 ENERGIFORSK

Energiforsk AB | Telefon: 08-677 25 30 | E-post: kontakt@energiforsk.se | www.energiforsk.se

Sammanfattning

Noggranna metoder för 3D- och 1D-beräkningar har byggts upp i projektet. Båda metoderna ger resultat som överensstämmer med de experimentella resultaten. Trycknivåerna överensstämmer med experimenten, eftersom hänsyn är tagen till förlusterna i ventilen.

Tryckoscillationerna som observerades i experimenten återfinns inte i de numeriska resultaten. En preliminär slutsats är att det inte är en våg i uppströmstanken som orsakar tryckoscillationerna utan att fluktuationerna kommer från fluid-struktur-interaktion mellan luckstängning och flödet i experimentet.

Summary

This work is a numerical study of the water flow in a square channel, during the closure of a gate. The flow is driven by a difference in surface elevation between an upstream and a downstream water tank. The case resembles, for example, the flow in hydro power water ways during a sudden closure of a turbine unit, in which it is of interest to quantify the amplitude and frequency of the following pressure waves.

This work is a continuation of the senior SVC project aiming at studying 1D-3D coupling and system transients coupled to local flow unsteadiness. That project presented initial experimental and numerical studies under the same conditions as in the present work. The current objective is to address the numerical modeling of transients with special focus on detailed 3D processes interacting with the flow in an essentially 1D geometry.

The numerical part is divided in three approaches, one-dimensional simulation of the pressure change in the channel, three-dimensional simulations of the flow details at the closing gate, and finally a coupled one-dimensional three dimensional approach. The last method combines the precision of the three dimensional modelling at places where the flow is complicated, and the simplicity and speed of the one-dimensional modelling for the pipe system. The simulations are time-resolved. Step one and two are presented in this work. Step three will be part of a second stage of the project.

The one-dimensional simulations are performed with a boundary condition for the flow rate at the gate, which is estimated from the experimental results. The results show a pressure variation, as a function of the closure of the gate, that behaves somewhat similar to the experimental results. The oscillating behaviour of the pressure observed in the previous experimental work is however not observed in the one dimensional simulations.

The 3D simulations are performed using a computational domain that includes the upper and lower tank and its free surfaces. A flow rate is specified both at the inlet and outlet of the system, and the free surfaces dictates the pressure level in the system. A lot of time was spent dealing with a flow regulation valve below the upper tank that generated a pressure loss in the system, which was unfortunately not taken notice of in the original experimental study. The flow is assumed to be incompressible, and the time dependent Navier Stokes equations are solved in the system. The closing gate is modelled by cutting up the computational mesh. The results show that both the flow and the pressure behave as in the experiment. The pressure levels are similar as those in the experiment, since the loss generated by the regulating valve is taken into account. The oscillations observed in the experiments are not present in the numerical results, and it is suggested that these fluctuations could come from fluid-structure interaction between the gate and the flow in the experiment.

Innehåll

1 Introduction

The role of hydropower to provide balance regulating power is important to the Swedish power system. This role is becoming even more accentuated with the expansion of other renewable electricity sources, i.e., mainly wind. The dynamics of hydropower ranges over a large spectrum of scales, from parts of a second to years. However, for scales larger than minutes flow may be considered as steady from a hydro-dynamical point of view. Transient terms of momentum are negligible compared to other terms, forces. In this study those shorter scales, below a few seconds, are being addressed. From a practical point of view these shorter scales are manifested mainly as pressure transients. These pressure transient need to be considered in the design and operation of hydropower plants. A review of all time-scales of hydropower is given by Dahlbäck (2011).

1.1 OBJECTIVE AND SCOPE

The objective is to address the mathematical modelling of rapid pressure transients with special focus on detailed 3D processes interacting with pressure transients travelling in essentially a 1D geometry. The limitations and benefits of 1D and 3D modelling will be outlined and the combination of 1D and 3D modelling assessed. One key component in the study is the comparisons with measurements in an experimental facility where high Reynolds number conditions could be provided. Here the gate closing downstream a long rectangular channel has been tested. This test constitutes the verification case for this study, both in 1D and 3D, and was presented in a previous Elforsk report (Cervantes et al. (2013)), as a result of a senior SVC project

1.2 PRESSURE TRANSIENTS

Rapid changes of flow conditions in hydraulic systems (i.e. flow rate) are accompanied by transients in pressure. The amount of pressure change induced by the flow on the ambient containment depends on velocity, geometry and material properties of both the fluid and the containment. In hydropower the most striking feature in this sense is perhaps the amount of moving mass in the system; usually large flow rates in large volumes. Considerations how to handle pressure transients are therefore an important feature of hydropower plant operations. Turbine, valve and gate regulations will cause pressure transients to occur. Since both the system of waterways often is complex and the operational changes may be done in a number non-independent sequences the hydraulic system dynamics could be very complex. Although operations usually have limitations regarding regulations resulting in transients, it is often virtually impossible to consider all combinations of events that could occur.

In hydropower two transient phenomena are usually of importance; rapid water hammer, and slow pressure surges. The prior, water hammer, fluid hammer or hydraulic shock, is caused by the compression of fluid volumes when retarded. The propagation of the pressure change is limited by the wave speed of the system. The period of such transients in hydropower is in the order of parts of a second. The latter phenomena, pressure surges, are caused by the dynamic interaction between momentum of the water mass and non-elastic pressure forces. The time scale of pressure surges is typically in the order of seconds.

1.3 PREVIOUS WORK

During the new-build period of hydropower in Sweden, limitations of operation was usually determined by physical models or in less complex cases by more or less calculations by hand. The basic theory of and methods to predict pressure surges has been well documented in textbooks since long, e.g. Fox (1989), Jaeger (1977), Tullis (1989), and Wylie & Streeter (1982).

However, since several decades ago computer models dominate the analysis of more complex pressure surges. Since one-dimensionality is a fairly accurate representation of most hydropower waterway systems, the complexity of analysis becomes quite limited. More complex components like bends, valves, gates, etc. may here be represented by forces of a semi-empirical origin. See for example textbooks like Miller (1990) and Idelchik (1996). Such 1D mathematical computation of transient flow phenomena have been used to address complex phenomena in sometimes complex geometries, e.g., Adamkowski (2001), Jiménez & Chaudhry (1987), Ramos & Almeida (2001), and Yu et al. (2011).

When the complexity of the involved flow components increase, full 3D modelling may be used. Today, several softwares exist for 3D computational fluid dynamics (CFD). To resolve the rapid transient phenomena extra care must be taken in the choice of discretization of governing equations (in time and space) and in the numerical scheme to solve these. Especially the pressure equation needs attention here. In this study we have chosen to use a free open source CFD software called OpenFOAM (www.openfoam.com). Although CFD-simulations are available today, a full 3D resolution of long, essentially one-dimensional, tunnels is not a motivated use of computational resources. The idea to combine 3D CFD in regions with complex flow with 1D models further away in the more uniform geometries is therefore tempting. Some efforts where this has been done may be found, e.g. Montenegro et al. (2007), Montenegro & Onorati (2009), and Zhang (2012).

The work done by Cervantes et al. (2013) in a senior SVC project is the base of the current work. An experimental test rig was designed and built to provide reliable experimental pressure and velocity measurements, and the flow was computed in a squared pipe using a 1D-2D numerical model in OpenFOAM. The numerical method showed promising results, but concluded as well of the importance of including a larger part of the system in the computational domain to accurately predict the behaviour of the flow.

2 Experimental apparatus and methods

2.1 WATER TUNNEL

The test rig and experimental campaign were achieved in a previous work. For more detail about it, see Cervantes et al. (2013).

The layout of the bottom outlet rig, as shown below in Figure 1, is mainly composed of water feeding system, an upstream and a downstream water tank, 10 m rectangular pipe and a control valve. During the experiment, the valve was nearly closed to allow a flow rate of 50 $L \cdot s^{-1}$. The rig is pictured in Figure 2.

The horizontal pipe is composed of five test sections, each being 2 m in length. The walls of the pipe are made from glass sheet, 8 mm in thickness. The inner pipe dimension measures 200 mm (width) by 250 mm (height). The section is placed in an aluminum frame to guarantee its structural integrity. The gate is made in sheet steel, see Figure 3. The valve is pneumatic and its control is incorporated into the pressure measurement system of the rig. The typical closing and opening time varies between 2 and 10 seconds.

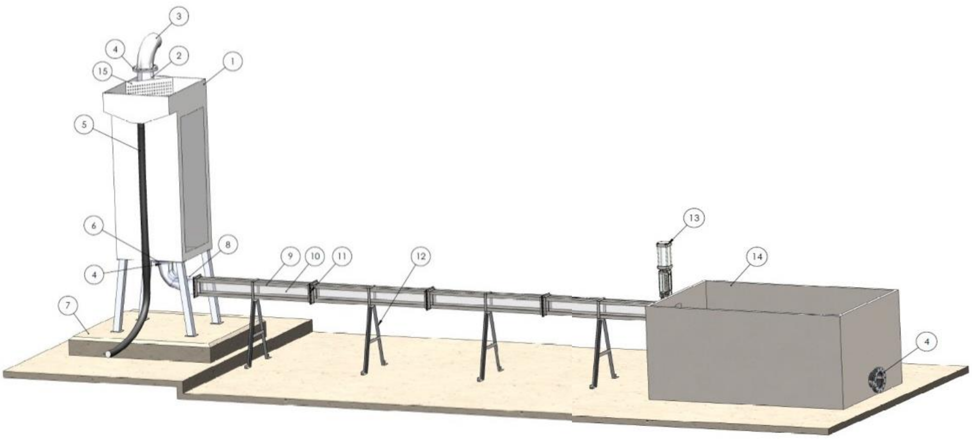


Figure 1 Layout of text rig.

Figure 2 Test rig seen from upstream.

Figure 3 Control valve of knife type of the rig and pressure taps position.

2.2 INSTRUMENTATION

Differential pressure sensors from Druck, PDCR 810, with two different measuring ranges (0-0.7 and 0-1 bar) were used. They have an accuracy of ±0.1%. The pressure taps were placed at cross-section 1 and 4 of them at cross-section 2. From the gate upstream, the first cross-section is situated in the middle of the closest pipe section, i.e., 1.245 m from the gate. The second section is situated 3.245 m from the gate further upstream. Each pressure tap was placed in the middle, see.The velocity measurements were performed with a particle image velocimetry system (PIV) from Lavision GmbH.

The PIV measurements were performed at 8 different planes starting below the gate and upstream. 3 planes were used in the vertical direction to investigate the entire flow, Figure 4.

The PIV system and data acquisition system recording the pressure signal and the gate position were coupled enabling simultaneous measurements of the pressure, gate position and velocity function of time.

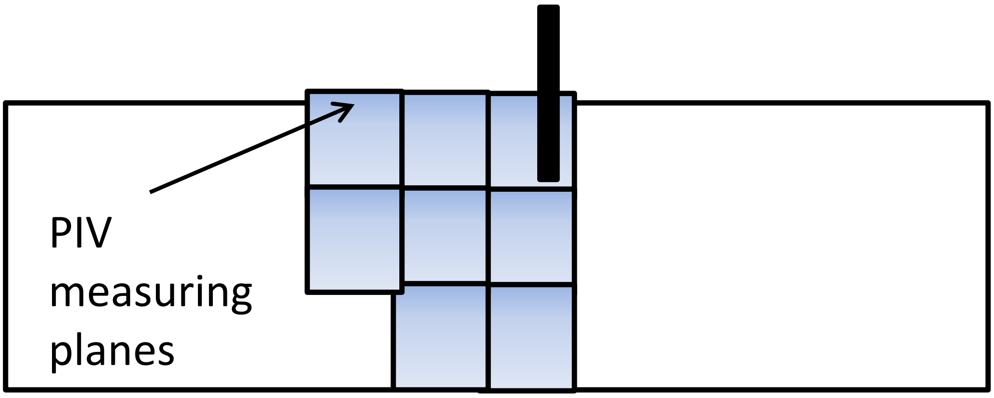


Figure 4 Schematic representation of the PIV measuring planes investigated during the measurements.

3 Numerical modelling

3.1 1D MODELLING

Simulink was used to model the system in the present work. Simulink is a developed by Mathworks and is integrated in the Matlab environment. It is a data flow graphical programming language tool for modelling and simulating dynamic systems. The whole system can be modelled using a series of block that represent the different parts of the systems (pipes, bend, gate, and so on). The present system includes the upper tank, the pipe system that leads the water to the gate, the gate and the lower tank. The system simulated in the present work can be seen in Figure 5.

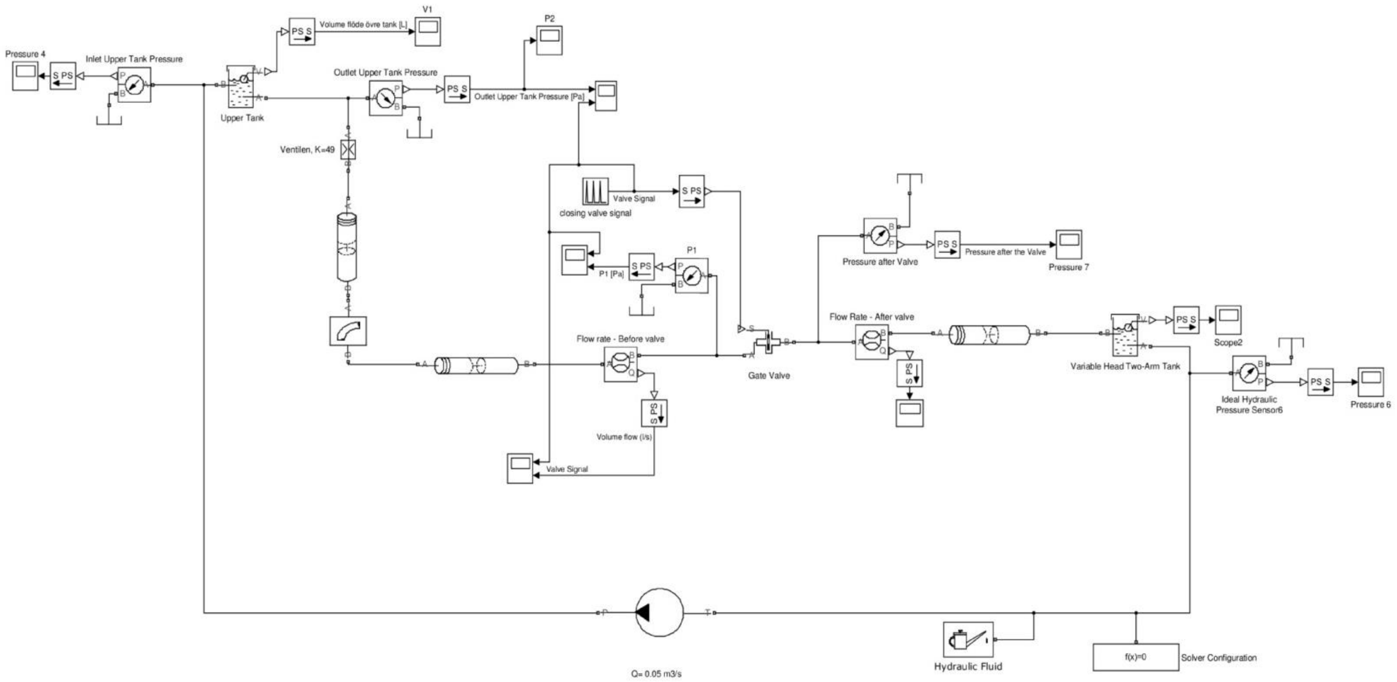


Figure 5 1D model of the present system. The system includes the upper tank, the pipe system, the gate and the lower tank.

3.2 3D MODELLING

Implementation details

Although the transient behavior of the flow, the water was assumed to be incompressible, to simplify the problem. The interDyMFoam OpenFOAM solver was used as a base for the work. It implements the laminar incompressible Navier Stokes equations, and controls the free surface in the upper and lower tanks through the Volume of Fluid method. The laminar assumption is made, thus no turbulence model is presently included. The closing valve is modeled using the dynamicFvMesh/linearValveLayersFvMesh OpenFOAM dynamic mesh livbrary. It features sliding interfaces with mesh topology changes at the mesh surfaces along both sides of the valve, and removal of mesh layers as the valve is moving downwards.

The computational domain of the 3D model is quite large. To be able to obtain results as fast as possible, the case needs to be parallelized, for the simulation to run in parallel. This was not implemented correctly in the beginning for the solver used in this case, and a lot of time was spent on making the case smoothly run in parallel. The solver is now fixed, and the case can be decomposed and run in parallel for faster results.

Different boundary conditions were compared to simulate the pressure loss generated by the flow regulation valve between the upstream tank and the square pipe. Ultimately, a solid obstacle solution was chosen, and the model includes a flow regulation valve that obstructs about 90% of the total area of the pipe below the upper tank, generating the pressure loss that can be seen in the experiment data.

Computational domain, mesh and valve motion

Figure 6 shows the computational domain, as well as some feature of the mesh. The mesh consists on 2 million of nodes, and is fully hexahedral, generated in ICEM CFD. The upper and lower tank free surfaces are included in the present domain. A General Grid Interface is used between the pipe system and the gate, to be able to replace the pipe with a 1D system in the future.

Figure 7 shows dynamic mesh motion of the gate. The gate is moving downwards with a constant velocity (0.05 m/s), removing cell layers and modifying the mesh topology.

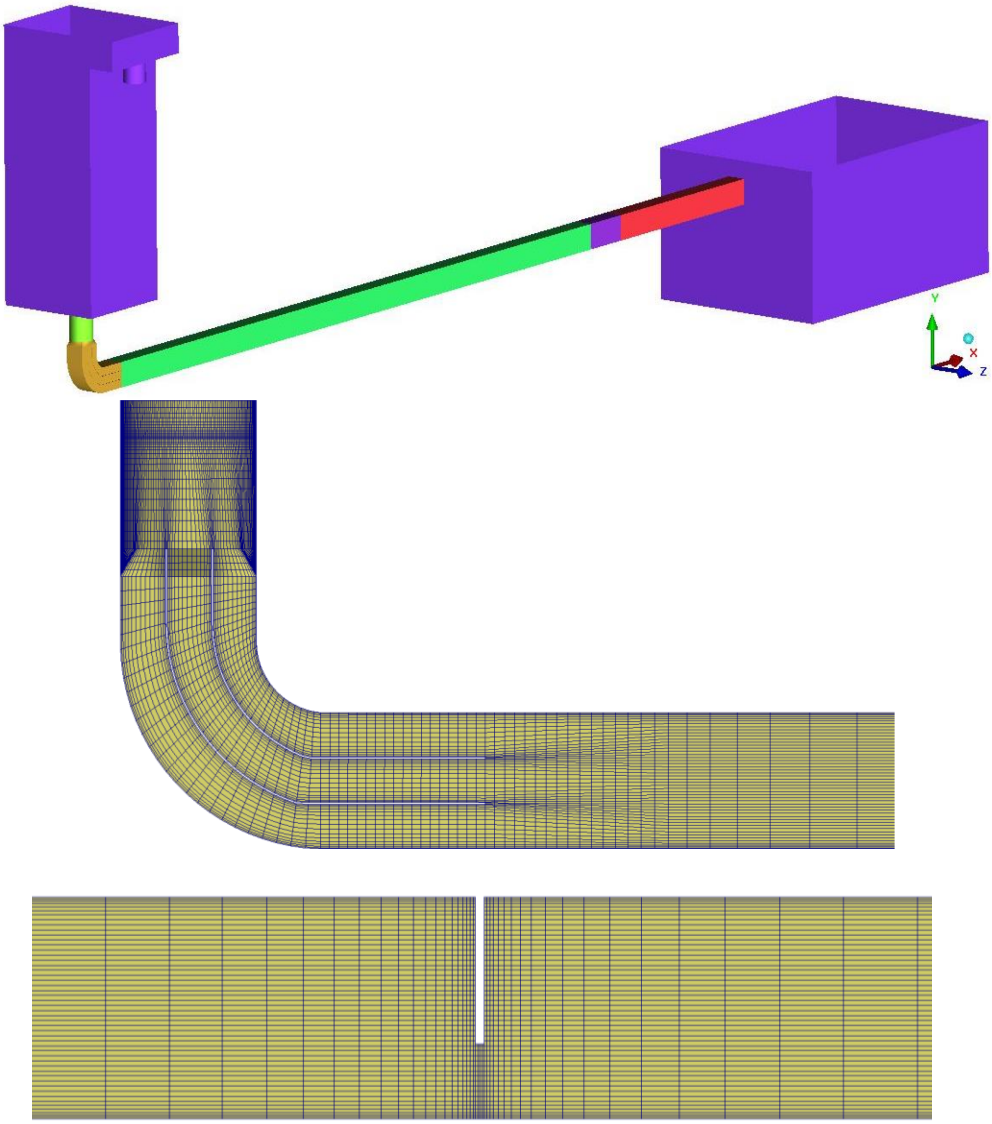


Figure 6 CAD model prepared for generation of CFD mesh in ICEM CFD.

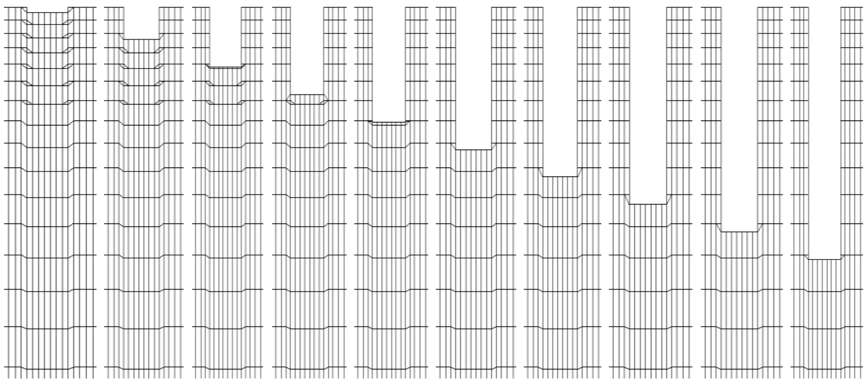


Figure 7 Dynamic mesh motion of the gate The figure shows a number of frames of a zoom-up of the valve as it is closing.

Case set-up and solution procedure

The inlet and outlet velocity was set as boundary condition for the water, based on the experimental volume flow (50 l/s) and on the inlet/outlet area. In a first time, the gate was kept open so that the flow would develop in the system, and the upper and lower free surfaces would reach a stable level. The closing valve simulation, with valve speed is $U_y = -0.05m\ s^{-1}$, was then started from the fully developed solution. The time step of the closing valve simulation was set to $\Delta t = 0.0002$ s and a maximum Courant number of Co = 0.2. The simulation terminated just before full valve closure, at 4.9 s, since at least one mesh cell must be preserved. The simulation continued from the results at 4.9 s, with the valve velocity set to zero. Finally, the valve would open again until it reached its start position.

4 Results

4.1 1D RESULTS

From the 1D-simulations it is clear that the main physics is captured. Figure 8 shows the flow behavior for the pressure before the gate. The increase in pressure during the valve closing seems to be correct, and while the gate is closed, the pressure increases due to the increase of water level in the upper tank. The compressibility of the water was not simulated here, thus the wave propagation frequency is not reproduced. The other frequency observed in the experimental data is not to be seen either.

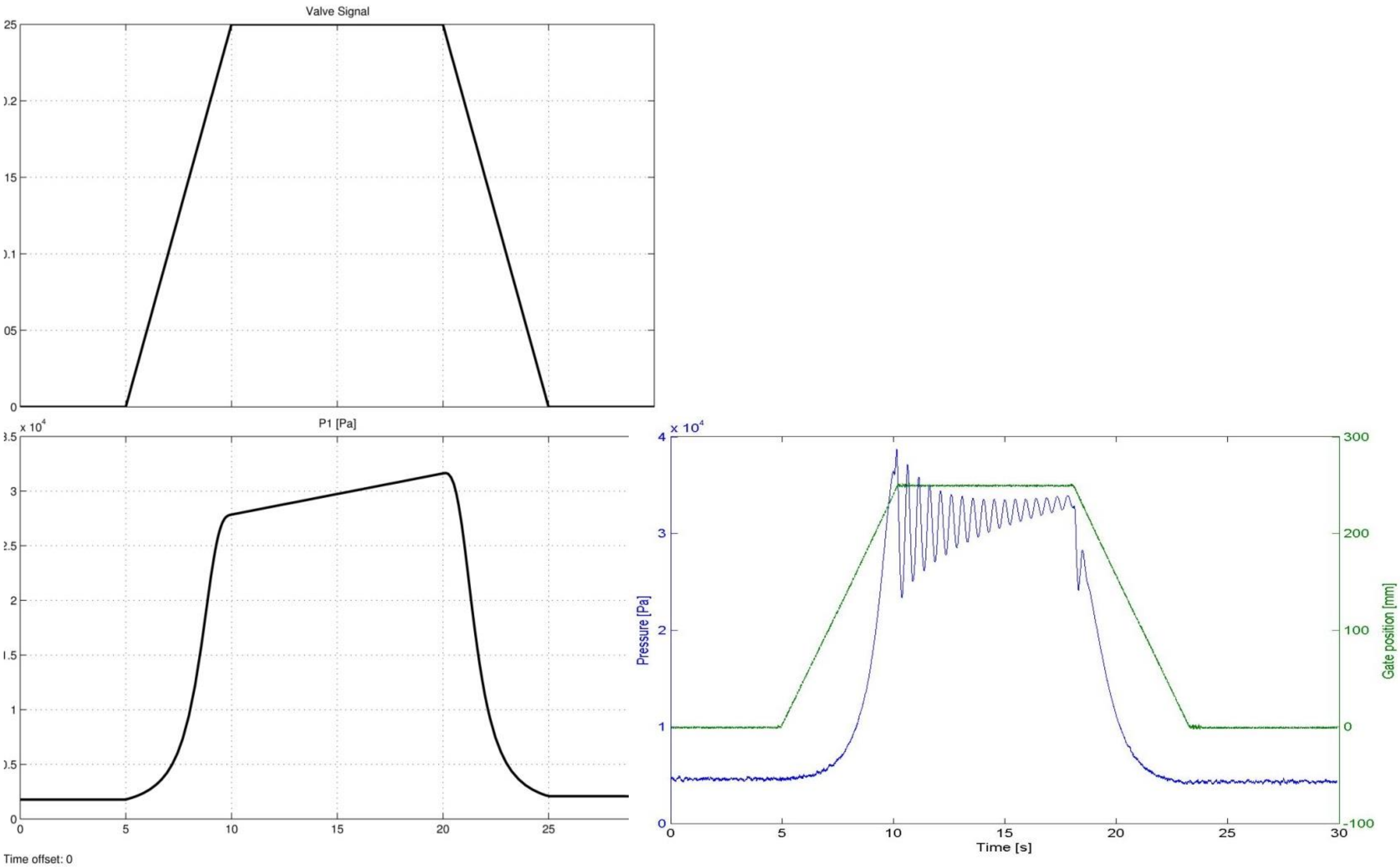


Figure 8 Pressure behavior of the water behind the valve during the closing and opening of the gate.

4.2 CFD RESULTS

At the end of the first stage of the present work the first preliminary results with somewhat proper boundary conditions, and without large unphysical disturbances, have been achieved. Further investigations regarding boundary conditions, parallelization validation need to be done in order to get more reliable results. The computational time for the closing valve phase in the full 3D non-parallelized case is 2 month. A lot of time was thus spent on fixing fix the parallelization issue.

Figure 9 shows the flow features when the flow has been developed, and before the gate starts closing. The gate then is closing, and Figure 10 and Figure 11 show the velocity and pressure when the gate is almost fully closed. A large static pressure builds up on the upstream side of the valve, as the flow is decelerated. Most of the static pressure build-up occurs at the final 20% of the valve closure time. Figure 11 shows that the flow is decelerated and that a large recirculation and a wall jet is formed after the valve. Figure 10 shows as well the impact of the flow regulation valve, under the upstream tank, on the pressure in the system.

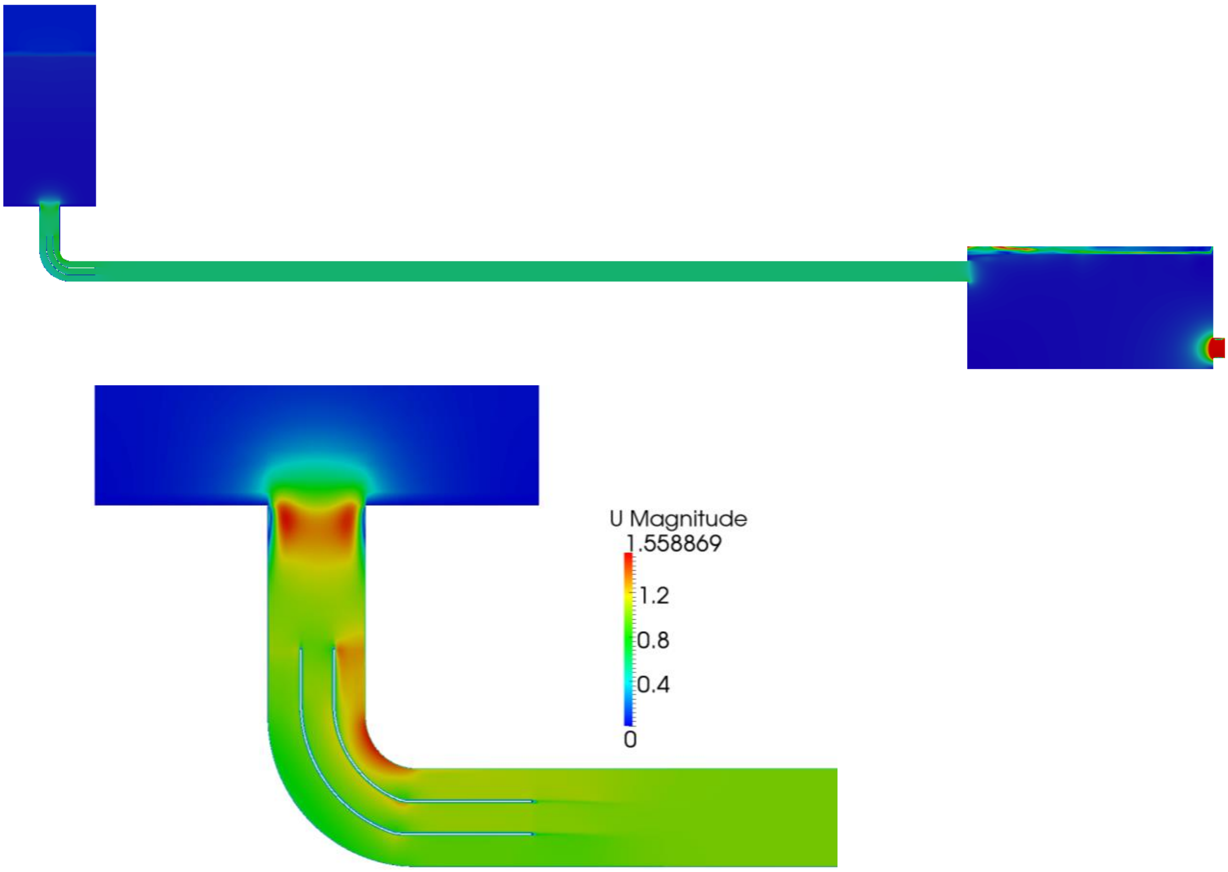


Figure 9 Snapshots of the velocity magnitude in the entire computational domain, and in a zoom-up at the bend, below the flow regulation valve, for the developed flow.

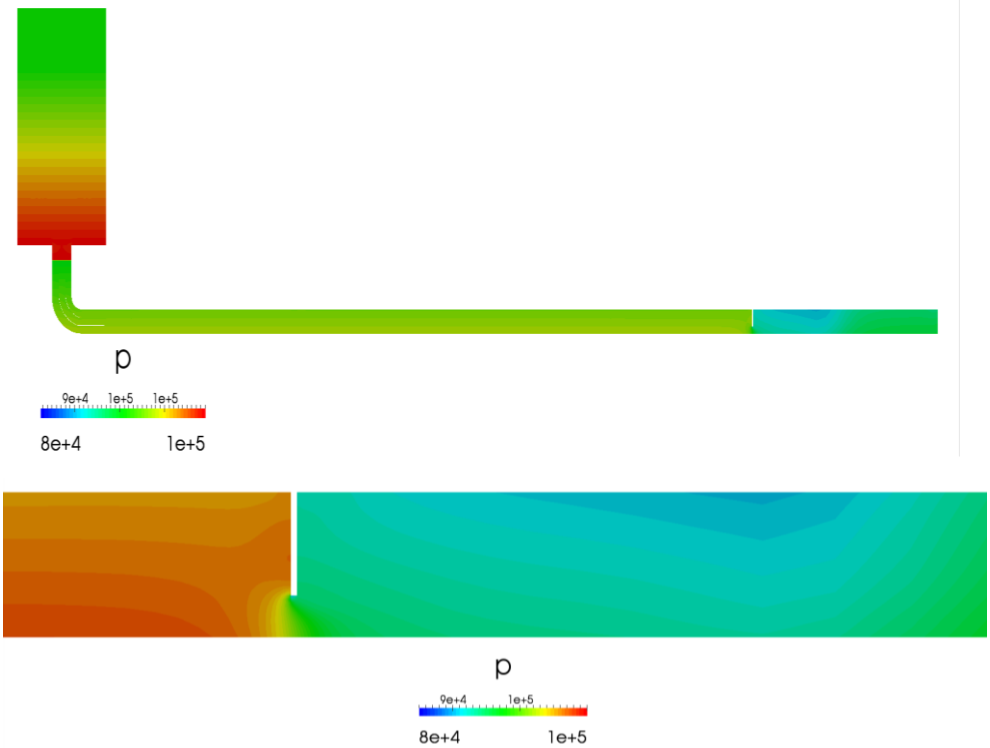


Figure 10 Snapshots of the pressure in the entire computational domain, and in a zoom-up at the valve.

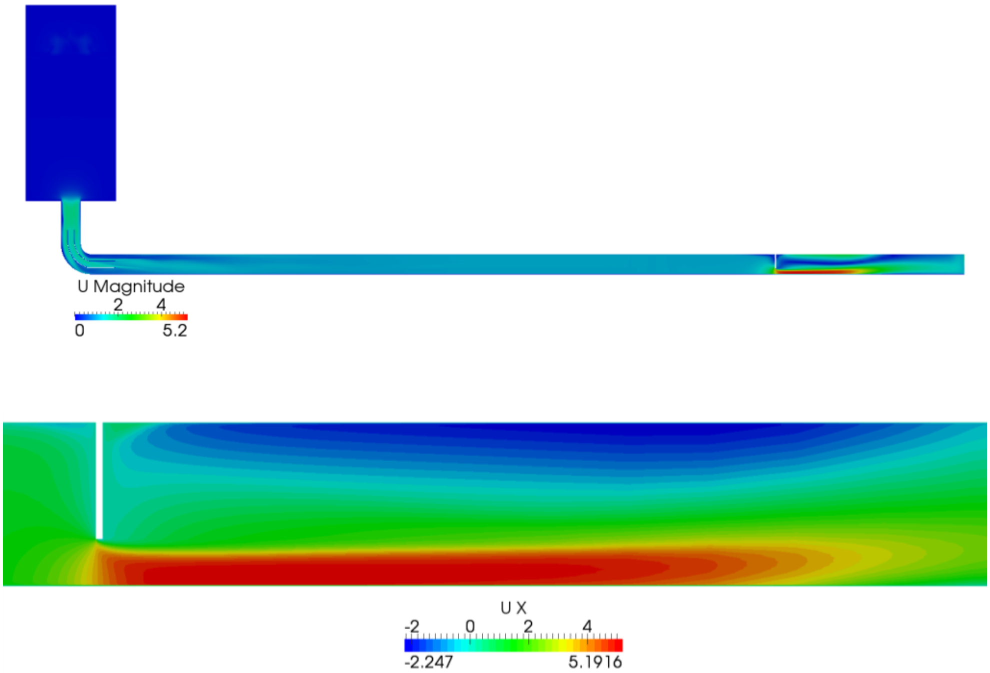


Figure 11 Snapshots of the pressure in the entire computational domain, and in a zoom-up at the valve.

5 Conclusion

The case is studied numerically with two approaches. It is found that the settings used in the 1D approach are insufficient to model the oscillations in the system, although the global pressure behaviour of the flow is correctly captured. This suggests that the physics governing the oscillating behaviour is not included in the numerical set-up. It was suggested in the previous work by Cervantes et al. (2013), that the oscillations have their origin in waves in the upper tank, but it may as well be due to waves in the lower tank in combination with fluid-structure interaction between the flow and the closing gate. The results from a 3D approach capture the increment in pressure as a function of gate closure quite well. The focus in the near future should be to finish validating the parallelization of such a case, as well as moving to a 1D-3D model approach.

6 References

Adamkowski, A., 2001, "Case study: Powerplant penstock failure", J. Hydraulic Eng., Vol. 27, pp. 547-555.

Dahlbäck, N. , 2011, "Utvecklingsbehov inom reglerkraftsområdet ur ett vattenkraftsperspektiv", Report 10:11, Elforsk/SVC.

Fox, J. A., 1989, "Transient , flow in pipes, open channels and sewers", Textbook, Ellis Horwood Ltd., Chischester, UK.

Idelchik, I.E., 1996, "Handbook of Hydraulic Resistance", 3rd ed., Begell House Inc., New York, NY, USA.

Jaeger, C., 1977, "Fluid transients", Textbook, Blackie & Sons Ltd., Glasgow, UK.

Jiménez, O. F., and M. H. Chaudhry, 1987, "Stability of hydroelectric power plants", J. Energy Eng., Vol. 113, pp. 50-60.

Jonsson, P. P., 2011, "Flow and pressure measurements in low-head hydraulic turbines", Doctoral thesis, Luleå University of Technology.

Kundu P., 1990, "Fluid mechanics", ISBN-10: 0124287700

Larock, Jeppson, & Watters, "Hydraulics of Pipeline Systems", 2000

Miller, D. S., 1990, "Internal Flow Systems", BHRA Information Services, Bedford, UK.

Montenegro, G., Onorati, A., 2009, "Modeling of Silencers for I. C. Engine Intake and Exhaust Systems by means of an Integrated 1D-multiD Approach", SAE Int. J. Engines 1(1): 466-479.

Montenegro, G., Onorati, A., Piscaglia, F., D'Errico, G., 2007, "Integrated 1D-MultiD Fluid Dynamic Models for the Simulation of I. C. E. Intake and Exhaust Systems", SAE Technical Paper 2007-01-0495.

Ramos, H., and A. B. Almedia, 2002, "Dynamic orifice model on waterhammer analysis of high or medium heads of small hydropower schemes", J. Hydraulic Res., V. 39, No. 2, pp. 429-436.

Tullis, J. P., 1989, "Hydraulics of pipelines - pumps, valves, cavitation, transients", Textbook, John Wiley & Sons Inc., New York, NY, USA.

Wylie, E. B., and V. L. Streeter, 1982, "Fluid transients", Textbook, Feb Press, Ann Arbor, MI, USA.

Yu, X., J. Zhang, and A. Hazrati, 2011, "Critical superposition instant of surge waves in surge tank with long headrace tunnel", Can. J. Civ. Eng., Vol. 38, pp. 331-337.

Zhang, X.-X., 2012, "Simulation of Hydraulic Transients in Hydropower Systems Using the 1-D-3-D coupling approach", J. Hydrodynamics, Ser. B., V. 24, No. 4, pp. 595-604.

Cervantes M., Andreasson P., Hellström G., Nilsson H., Yang J.," Experimental and numerical investigation of a hydraulic channel during a gate closure at a high Reynolds number", Final report of a project in SVC, 2013.

1D SYSTEM TRANSIENTS COUPLED WITH 3D LOCAL FLOW DETAILS