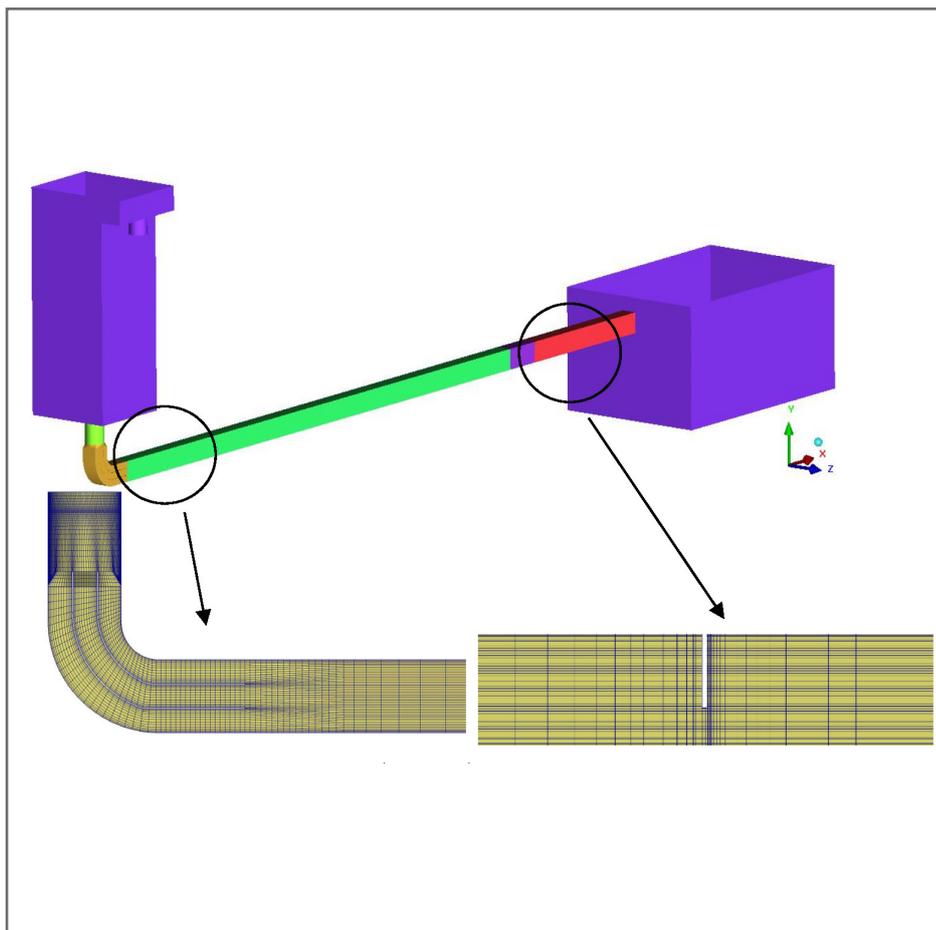
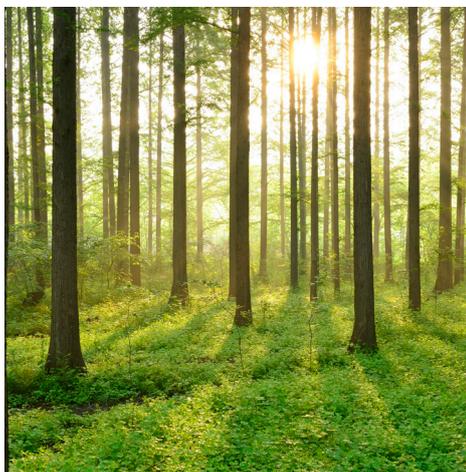


1D SYSTEM TRANSIENTS COUPLED WITH 3D LOCAL FLOW DETAILS

REPORT 2016:332



1D System Transients Coupled with 3D Local Flow Details

Report after 2nd stage

OLIVIER PETIT

ISBN 978-91-7673-332-5 | © 2016 ENERGIFORSK

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

Foreword

Det här är rapporten från ett postdoktoralt projekt vid Chalmers. Projektet har bedrivits inom Svenskt Vattenkraftcentrum, SVC som är ett centrum för utbildning och forskning inom vattenkraft och gruvdammar. Syftet är att säkerställa kunskapen för en effektiv och tillförlitlig vattenkraftproduktion och en trygg dammsäkerhet i Sverige.

Svenskt Vattenkraftcentrum har etablerats av Energimyndigheten, Energiforsk och Svenska Kraftnät tillsammans med Luleå tekniska universitet, Kungliga Tekniska Högskolan, Chalmers tekniska högskola och Uppsala universitet.

Medverkande organisationer är: Andritz Hydro, Falu Energi & Vatten, Fortum Generation, GE Renewable Energy, Holmen Energi, Jämtkraft, Jönköping Energi, Karlstads Energi, Mälarenergi, Norconsult, Rainpower, Skellefteå Kraft, Sollefteåforsens, Statkraft Sverige, Sweco Energyguide, Sweco Infrastructure, SveMin, Umeå Energi, Uniper, Vattenfall Research and Development, Vattenfall Vattenkraft, VG Power, WSP Samhällsbyggnad och ÅF Industry.

Reported here are the results and conclusions from a project in a research program run by Energiforsk. The author / authors are responsible for the content and publication which does not mean that Energiforsk has taken a position.

Sammanfattning

Detta arbete är en numerisk studie av vattenströmningen i en fyrkantig sluten kanal, i samband med stängning av en ventil. Vattenströmningen drivs av nivåskillnaden mellan vattenytorna i en uppströmstank och en nedströmstank. Fallet efterliknar exempelvis strömningen i vattenkrafttunnlar vid plötslig stängning av ett aggregat, vid vilket det är av intresse att kvantifiera tryckvågornas storlek och frekvens i systemet.

Studien är uppdelad i tre angreppssätt. Tredimensionella inkompressibla beräkningar av strömningar i hela systemet, endimensionella kompressibla beräkningar av tryckförändringen i kanalen, och implementering och validering av en 1D-3D-kopplingsmetod som kombinerar de två först nämnda metoderna.

De tredimensionella beräkningarna utförs för hela systemet, från övre tanken till den nedre, och vattnets fria ytor tas med i beräkningarna. Vattnet antas vara inkompressibelt för att förenkla beräkningar. Detta är utfört för att undersöka om den tryckfluktuation som observerats i tidigare mätningar beror på fluktuationer i vattennivån i den övre behållaren när ventilen är stängd. Resultatet visar att det inte skapas någon tryckvariation i systemet när ventilen är stängd, och att den uppmätta tryckvariationen därmed inte beror på vågor på de fria vätskeytorna.

De endimensionella beräkningarna utförs med hjälp av randvillkor för flödet vid den stängande ventilen, uppskattat från de experimentella resultaten. Resultatet visar på en tryckvariation som beter sig på ungefär samma sätt som i experimentet. Dock är nivån på förändringen inte lika stor som i experimentet. Vid fullt stängd ventil uppstår en fluktuation med avtagande amplitud, som dock är mer högfrekvent än i mätningarna och sker runt ett annat medelvärde än i mätningarna. Frekvensen på denna fluktuation motsvarar en ljudutbredningshastighet som modifierats för att ta hänsyn till mängden luft i vattnet, i kombination med kanalens längd.

De kopplade 1D-3D-beräkningarna utförs genom att beräkna 1D-delar av systemet med hjälp av "Methods Of Characteristics" (MOC), medan 3D detaljer beräknas med hjälp av en kompressibel lösare i OpenFOAM. Resultaten visar att både flöde och tryck beter sig som i experimentet. Trycknivåerna är liknande de i experimentet tack vare att hänsyn tagits till inloppets engångsförlust och mängden luft i vattnet. Vid fullt stängd ventil uppstår en tryckfluktuation som är mycket likt den experimentella fluktuationen.

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.

First the pressure oscillations observed during the measurement campaign when the gate is closed is analyzed. 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 observed in the measurement campaign are indeed pressure transients occurring due to the water hammer effect.

Secondly, the upper tank and the pipe are modelled using a compressible 1D code based on the method of characteristics (MOC). A comparison with the experimental data shows that the correct unsteady behavior of the system is captured by the 1D approach if the losses and the gate characteristics are correctly accounted for, at the same time as the compressibility is adapted to the air contents of the water and flexibility of the structure

Finally, a coupled 1D-3D method is introduced, that combines the precision of the three-dimensional modelling in regions where the flow is complicated, and the simplicity and speed of the one-dimensional modelling for the pipe system. The simulations are time-resolved. The results obtained with the 1D-3D coupling method are in very good agreement with the experiments, and show that the method works.

List of content

1	Introduction	9
1.1	Objective and scope	9
1.2	Pressure transients	9
1.3	Previous work	10
2	Test rig and experimental investigations	12
2.1	Design of test rig	12
2.2	Experimental setup	12
3	3D numerical investigations	13
3.1	Computational domain, mesh and gate motion	13
3.2	Case set-up and solution procedure	13
3.3	Results of the 3D numerical simulation	14
4	1D numerical investigations	16
4.1	Estimation of sound speed	16
4.2	Computational domain	16
4.3	Case set-up	17
4.4	Results of the 1D numerical simulations	17
5	Validation of the coupling method	19
5.1	1D-3D coupling method based on Riemann invariants	19
5.2	Validation of the 1d-3d coupling method	19
5.3	Influence of coupling configuration	20
5.4	Influence of time discretization scheme	20
5.5	Influence of 3D mesh size	21
5.6	Application to the test rig	21
6	Conclusions	23
7	References	24

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 intermittent renewable electricity sources, such as wind power. The dynamics of hydropower ranges over a large spectrum of scales, from parts of a second to years. However, for scales larger than minutes the flow may be considered as steady from a hydro-dynamical point of view. In this study the 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 a special focus on detailed 3D processes interacting with pressure transients travelling in an essentially 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 software 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 the importance of including a larger part of the system in the computational domain to accurately predict the behaviour of the flow. Petit (2015) numerically investigated the test rig. Previous results show that the 3D incompressible method and the 1D compressible method give reasonable results, and conclude that combining both would give even better results at a fraction of the computing time.

In the present work, the OpenFOAM open source CFD tool is used to study a 3D incompressible flow in the system. A minor loss is estimated and implemented in the 3D model to get the same pressure loss in the system as the one observed in the experiments. The free surfaces of the upstream and downstream tanks are included

in the simulation to investigate their influences on the pressure fluctuations. In a second stage a 1D approach that takes into account a finite sound propagation speed is used to model the system. The minor pressure loss previously determined is implemented in the 1D model as well. The results show that the compressibility of the water and its mixture with air, as well as the flexibility of the structure, are likely to be the cause of the pressure fluctuations observed in the experiments. Finally, a novel 1D-3D coupling method is implemented, based on the characteristic equations and Riemann invariants. The 1D MOC code is implemented as boundary conditions in the OpenFOAM open source finite volume CFD software. The accuracy and robustness of the coupling method are verified by a simple water hammer case, with respect to the time differencing scheme and the mesh size ratio across the interface. When validated, the coupling method is then applied to model the pressure surge caused in the test rig, and the results are compared with experimental data.

The 3D and 1D numerical results have been presented and published at the “IAHR workshop group meeting on cavitation and dynamic problems in hydraulic machinery and systems” in Ljubljana, Slovenia in 2015 by Petit et al. (2015). The 1D-3D coupling method was developed in collaboration with a visiting Chinese PhD student, Chao Wang, who was supervised to perform the coupling method development. The coupling method is submitted for journal publication.

2 Test rig and experimental investigations

The test rig shown in Figure 1 was built at Vattenfall R&D Laboratory. The flow is driven by the difference in elevation between the water surfaces in the upstream and downstream water tanks.

2.1 DESIGN OF TEST RIG

The test rig and experimental campaign were achieved in a previous work. For more details about it, see Cervantes et al. (2013). The layout of the bottom outlet rig, as shown in Figure 1, is mainly composed of a water feeding system, an upstream and a downstream water tank, a 10 m rectangular pipe and a control valve. During the experiment, the upstream valve was nearly closed to allow a maximum flow rate of 50 l·s⁻¹.

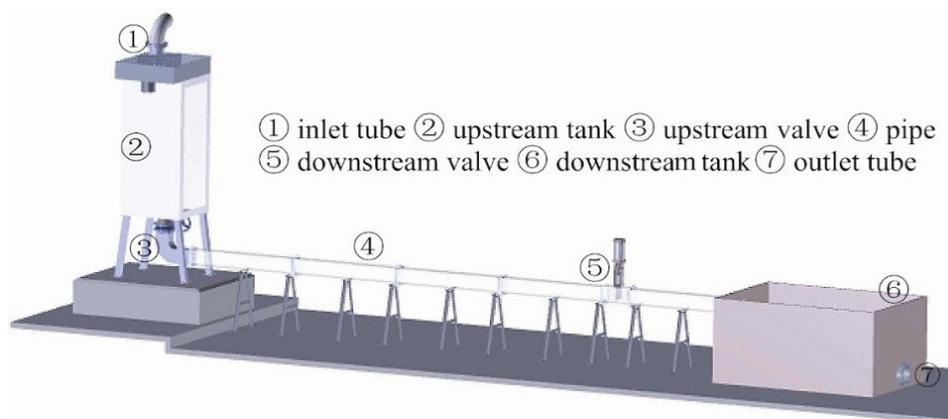


Figure 1: Layout of test rig

2.2 EXPERIMENTAL SETUP

During the experiments, the flow rate was maintained at a constant value of 50 l·s⁻¹, yielding a Reynolds number $Re=1.93 \cdot 10^5$ based on the hydraulic diameter $D_h=222$ mm and a water temperature of 15°C. 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 $\pm 0.1\%$. The pressure taps were placed at cross-section 1 (hereafter referred to as "S1", 1.245 m upstream the gate) and at cross-section 2 (hereafter referred to as "S2", 3.245 m upstream the gate). Velocity measurements were performed with a particle image velocimetry system (PIV) from LaVision GmbH.

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

3 3D numerical investigations

The experimental results raise the question of the pressure fluctuations during the period when the gate is closed. To understand if the free surfaces have an influence on the pressure fluctuations, the test case is numerically investigated using CFD. The OpenFOAM solver used to compute the flow implements the incompressible Navier-Stokes equations, and includes the free surface in the upper and lower tanks through the Volume Of Fluid (VOF) method. The assumption of incompressibility isolates the effects of the free surfaces by removing the pressure wave propagation.

3.1 COMPUTATIONAL DOMAIN, MESH AND GATE MOTION

Figure 2 shows the computational domain, as well as some feature of the mesh. The mesh is generated in ICEM CFD and consists on 2 million hexahedral cells. The upper and lower tank free surfaces are included in the domain. Two baffles in the bend are resolved. The knife valve below the upstream tank is modeled as a thin baffle that obstructs about 90% of the cross-sectional area of the pipe, generating a pressure loss that corresponds to that estimated from the experimental data in section 2.4. Figure 3 shows the dynamic mesh motion of the gate.

3.2 CASE SET-UP AND SOLUTION PROCEDURE

The experimental flow rate, $50 \text{ l}\cdot\text{s}^{-1}$, was used to set both the inlet and outlet velocity of the water to/from the upper/lower tanks. The total amount of water in the system is thus kept constant, but the head at the gate is allowed to change due to the varying free surfaces. To start with, 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 transient is achieved by closing the gate with a constant speed during 4.9 s (preserving a cell between the gate and the bottom boundary), keeping the gate closed 8 s, and opening the gate with a constant speed during 4.9 s. The time step was set to $\Delta t = 0.0002 \text{ s}$, yielding a maximum Courant number of $Co = 0.2$.

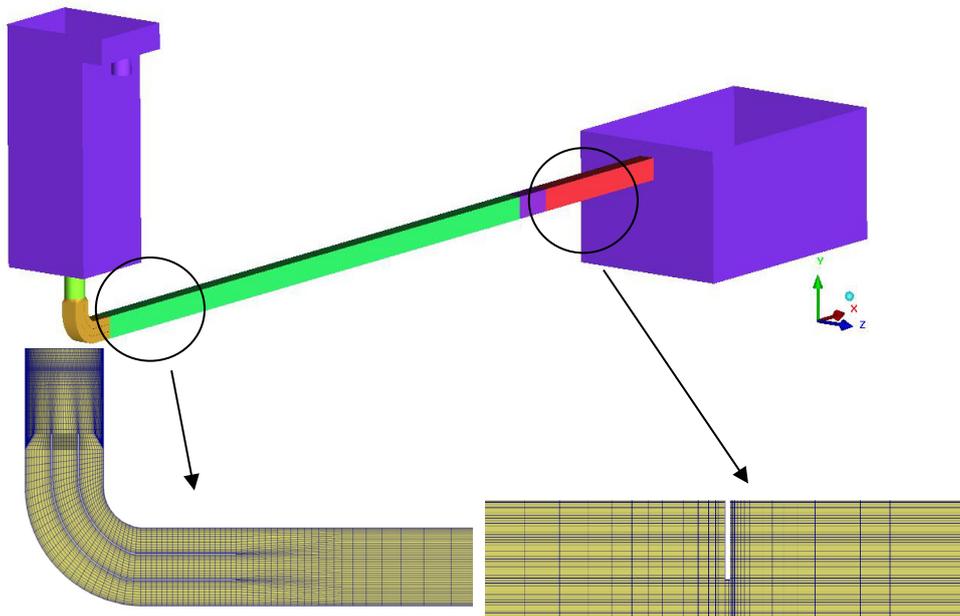


Figure 2: Computational domain and some features of the mesh.

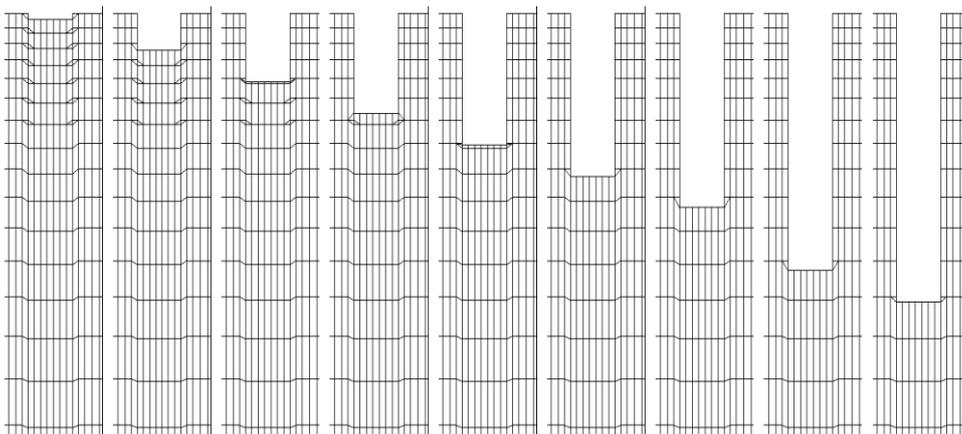


Figure 3: Zoom-up of the mesh at the gate as the gate is closing.

3.3 RESULTS OF THE 3D NUMERICAL SIMULATION

Figure 4 shows the pressure and velocity responses computed by the 3D unsteady incompressible solver. The transient starts at 5 s, after a stabilization period, followed by 4.9 s closing, 8 s closed and 4.9 s opening gate. The pressure increase is about $3.5 \cdot 10^4$ Pa in both the experimental and numerical cases, showing that the main behavior of the flow is well captured. When the gate approaches the fully closed position the overshoot of pressure is similar as in the measurement. For the experimental case this is the origin of the pressure wave. For the 3D computation the incompressibility assumption forces the pressure to immediately drop to the hydrostatic pressure. During the period when the gate is closed the pressure increases linearly due to the water level increase in the upstream tank, similar to what is observed experimentally. There are no fluctuations in pressure when the gate is closed, suggesting that the free surfaces are not the cause of the pressure fluctuations observed in the experimental results. When the gate is closed it can be

observed that the velocity at the gate is not totally zero, due to a leakage flow below the gate. The numerically predicted axial velocity variation along a vertical line 50 mm upstream the gate compares very well with the experimental data, see Figure 5.

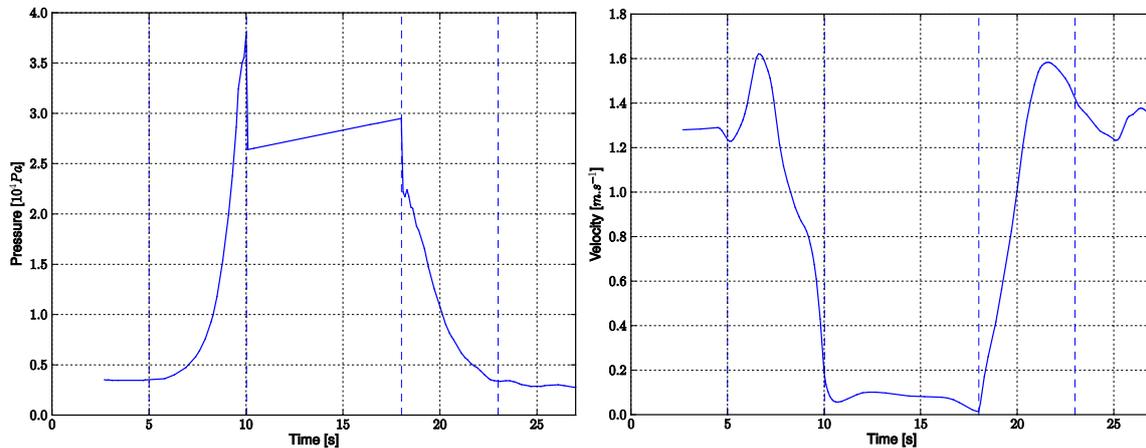


Figure 4: 3D computed pressure (left) and velocity (right) upstream the gate.

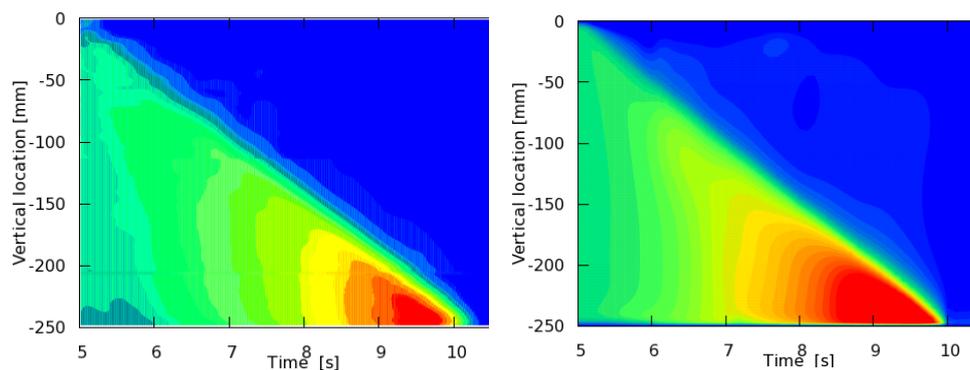


Figure 5: Measured (left) and computed (right) variation of the axial velocity along a vertical line 50 mm upstream the gate in the middle of the pipe as a function of time.

4 1D numerical investigations

To understand if the sound propagation speed has an influence on the pressure fluctuations, the test case is numerically investigated using Methods of Characteristics (MOC), a mathematical technique for solving hyperbolic partial differential equations.

4.1 ESTIMATION OF SOUND SPEED

The speed of sound in pure water at 25°C is about 1500 m.s⁻¹. However, the oscillations observed during the measurement campaign, in combination with the length of the pipe, suggest that the current speed of sound is approximately 80 m.s⁻¹. Wylie and Streeter (1982) show that the speed of sound can be reduced to the order of 80 m.s⁻¹ at a water temperature of 20°C and a 0.1% volume fraction of air. In the test rig, the air entrainment due to the pumping caused problems when the particle image velocimetry method (PIV) was used. There were too many, and too large, air bubbles in the water. The test rig was modified in order to reduce air entrainment to a minimum, but the problem was not eliminated. The air bubbles were present during the pressure measurements, but reduced before the PIV measurements. It is thus likely that the air content was high enough during the pressure measurements to give a speed of sound in the order of 80 m.s⁻¹. The structure of the pipe is as well not perfectly rigid, which also significantly reduces the sound propagation speed. The flexibility of the structure is unknown, but certainly contributes to the low speed of sound in the system.

4.2 COMPUTATIONAL DOMAIN

A reservoir-pipeline-gate system is used to compute the 1D behavior of the flow, see Figure 6. The system is divided into two parts. The first part of the computational domain includes the reservoir, its free-surface, and the vertical pipe. The second part of the domain models the horizontal pipe. The experimentally estimated minor-loss is implemented between the two parts of the domain to take into account the losses induced by the knife valve and the 90° bend. The dynamic wall friction losses are however not implemented in the present work. The grid size is 0.08 m, based on the sound speed and the time step to keep the Courant number at one.

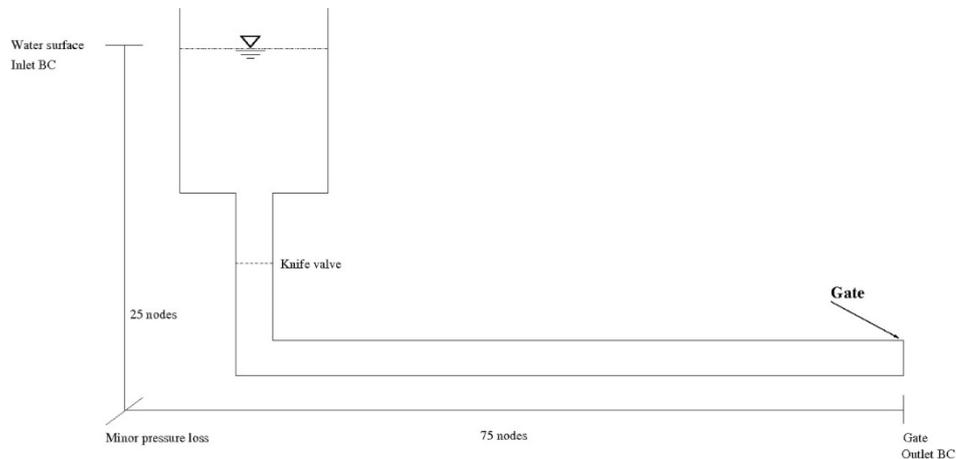


Figure 6: Computational domain for the 1D MOC computations.

4.3 CASE SET-UP

The time step for the 1D computation is set to $\Delta t = 0.001$ s. At the inlet an initial head of $H=3$ m is specified, as well as the discharge of $Q=0.05$ m³.s⁻¹. At the outlet, the time-evolution of the discharge is specified in different ways to mimic the characteristics of the gate valve.

4.4 RESULTS OF THE 1D NUMERICAL SIMULATIONS

When the gate closes the behavior of the pressure is dictated by the characteristics of the gate. Figure 7 illustrates this using three different discharge evolutions at the outlet boundary in the MOC computation. The pressure response at the gate is compared with the experimental signal. The elliptic discharge characteristic is the implementation that best fits the experimental pressure signal, but it still over-predicts the overshoot of pressure when the gate is closed. The gate discharge characteristic is instead determined from the pressure measurements during the gate closure. Figure 8 shows the calculated discharge at the gate when setting the extracted pressure as the outlet boundary during the gate closure. Figure 9 shows the computed pressure development using the calculated discharge from Figure 8 during the gate closure as outlet boundary condition, and assuming a no-flow condition when the gate is closed.

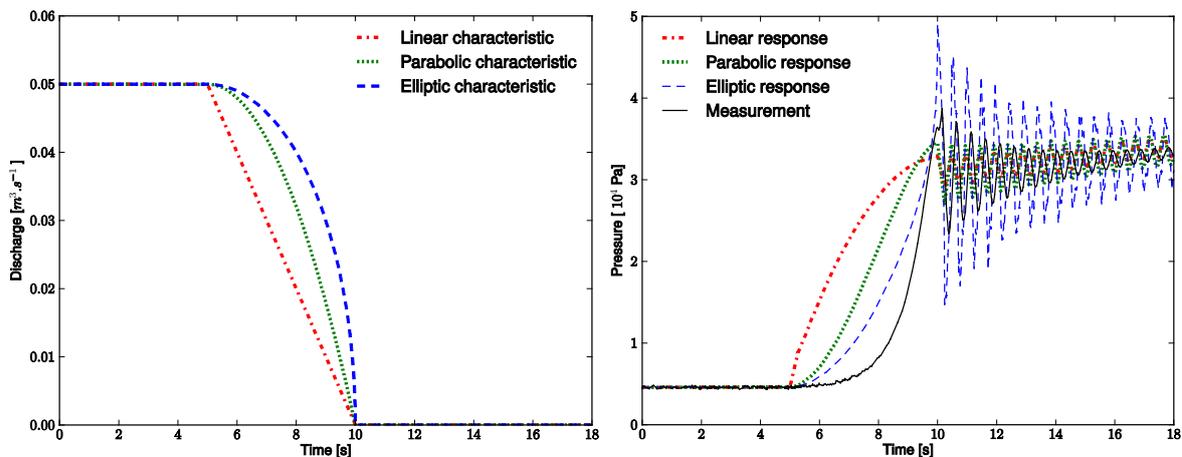


Figure 7: Approximate gate closing characteristic (left) and computed pressure response compared with the measurements (right).

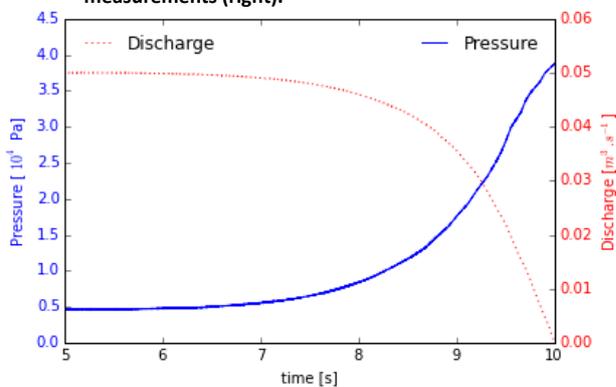


Figure 8: Measured pressure and corresponding approximated discharge, as the gate closes.

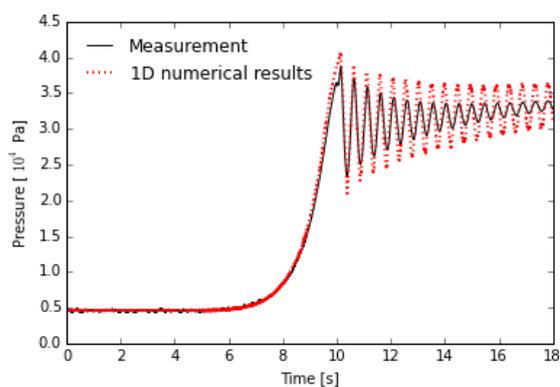


Figure 9: Pressure variations using the experimentally determined gate characteristics.

5 Validation of the coupling method

5.1 1D-3D COUPLING METHOD BASED ON RIEMANN INVARIANTS

Figure 10 shows a MOC-CFD-MOC coupling configuration for a duct. The regions between cross-sections 0-4 and 8-12 are 1D MOC zones, while the region between cross-sections 4-8 is a 3D CFD zone. MOC calculates the values at the cross-sections, while CFD calculates the values at the center of the cross-sections (cell-centered approach, here visualized in 2D). The boundary conditions are here set for MOC at cross-sections 0 and 12, using the velocity or pressure head, or a characteristic relation equation.

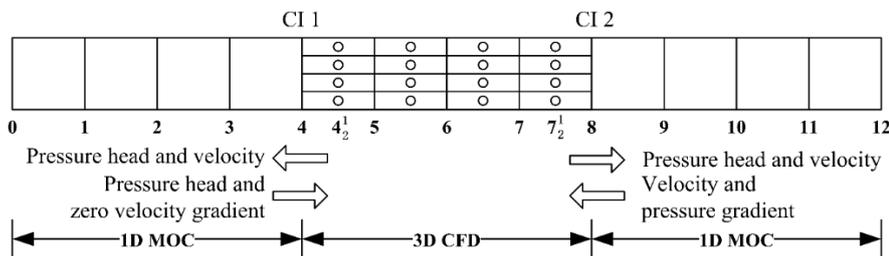


Figure 10: MOC-CFD-MOC coupling configuration for a duct

5.2 VALIDATION OF THE 1D-3D COUPLING METHOD

The 1D-3D coupling is validated using a case corresponding to the reservoir-duct-valve system shown in Figure 11. The duct is 1000 m long and has a square cross-section area of 1 m², and the sound speed is set to 1000 m/s. The reservoir is assumed to have an infinite surface area, so that a constant static pressure boundary condition corresponding to 100 m head can be set at the entrance of the duct. The initial discharge in the duct is 0 m³/s and the transient process is caused by a linear increase of the duct outlet discharge from 0 m³/s to 1 m³/s in 5 s, representing an opening of a valve. The duct wall boundary for CFD uses a slip condition and the friction resistance for MOC is zero in order to keep the same conditions for all the configurations. The time step for all the calculations is 0.005 s and the mesh size along the duct is 5 m for both the 1D and 3D regions.

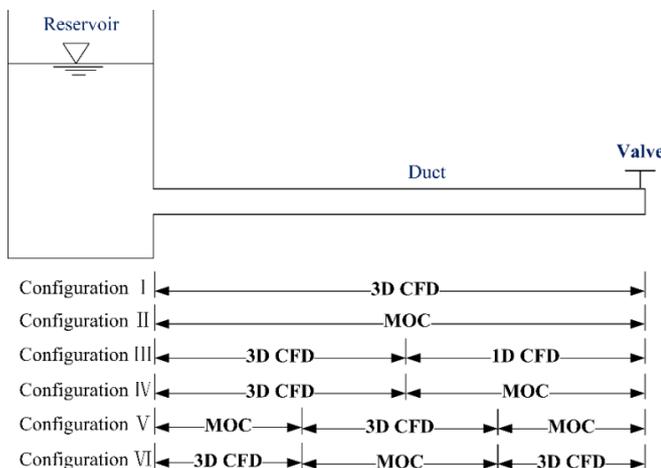


Figure 11: A reservoir-duct-valve system for validation

5.3 INFLUENCE OF COUPLING CONFIGURATION

The influence of different coupling configurations is investigated by comparing the results with the uncoupled cases using pure 3D CFD (Configuration I) and pure 1D MOC (Configuration II), as can be seen in Figure 11. At the same time it is verified that the two uncoupled methods give the same results. An additional base case for the validation is provided by Configuration III, which calculates half the domain with 3D CFD and the other half with 1D CFD using a general grid interface (GGI) for the coupling of those regions. The coupling method of the present work is applied to three configurations, named Configuration IV-VI in Figure 11. Configuration IV calculates half the domain with 3D CFD and the other half with 1D MOC. Configuration V couples two 1D MOC regions with one 3D CFD region, and Configuration VI couples two 3D CFD regions with one 1D MOC region. Figure 12 shows that the time varying pressure head at the valve is very similar for all the configurations. The configurations predict some different behavior at the extremum, but within 0.6% of the total variation. The inclusion of 3D CFD results in smoother curves and faster attenuation.

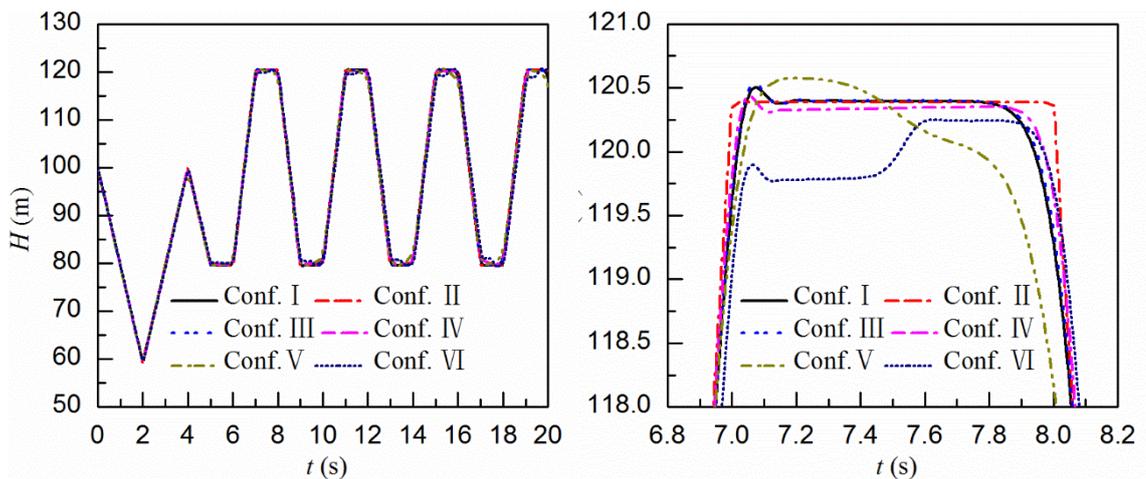


Figure 12: Pressure head at the valve for the different configurations. Left: overview, right: zoom-up of first peak

5.4 INFLUENCE OF TIME DISCRETIZATION SCHEME

The time discretization scheme influences the results of water hammer simulations, and in particular the attenuation of the pressure wave. The results using the first-order Euler, second-order Crank-Nicholson, and second-order backward schemes are therefore compared. Figure 13 shows the pressure head at the valve with different time schemes for Configuration I. It is shown that the result using the Euler scheme smears out the sharp edges and reduces the amplitude of the pressure pulsations. The pure Crank-Nicholson scheme, which is second-order when the coefficient in OpenFOAM is equal to 1, gives an overshoot that increases for each period. When the Crank-Nicholson coefficient is equal to 0 the scheme becomes identical with the Euler scheme. Here we use a coefficient of 0.45.

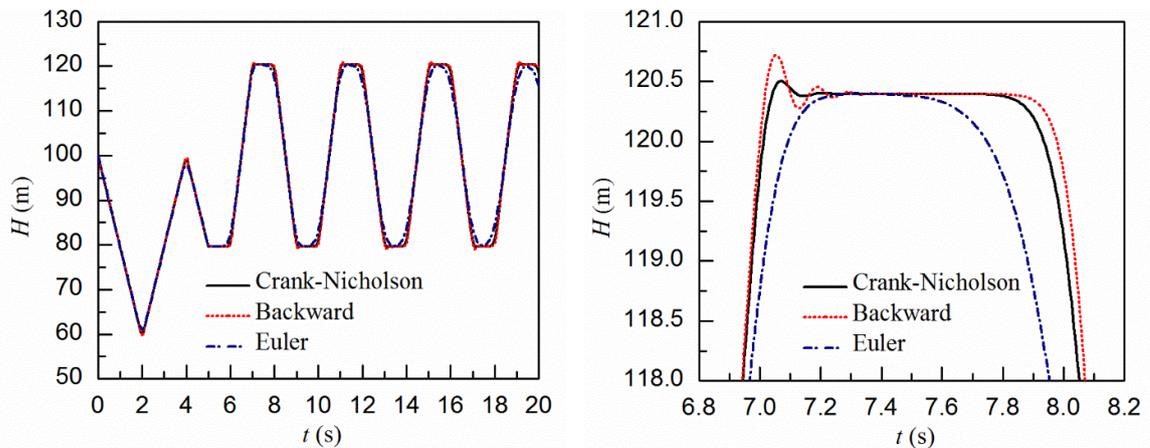


Figure 13: Comparison of results using Crank-Nicholson, Euler and Backward schemes. Left: overview, right: zoom-up of first peak

5.5 INFLUENCE OF 3D MESH SIZE

The spatial discretization in the 1D MOC regions is directly determined by the sound speed and the time step. The validation cases described in section 3.1 and 3.2 have a 5 m mesh size along the duct both for the 1D and 3D regions, to keep a pressure wave Courant number of 1 with a time step of 0.005 s. However, for real applications the 3D mesh is denser. It is thus important that the mesh size ratio across the interface does not influence the results. Figure 14 shows a comparison between the results of Configuration IV when the 1D MOC regions keep the original mesh distribution while the 3D CFD region varies the mesh size along the duct from 5 m to 2.5 m, 1 m, 0.5 m, and 0.25 m. It can be seen that both the size of the 3D mesh and the sudden change in mesh size at the interface have no significant influence on the results.

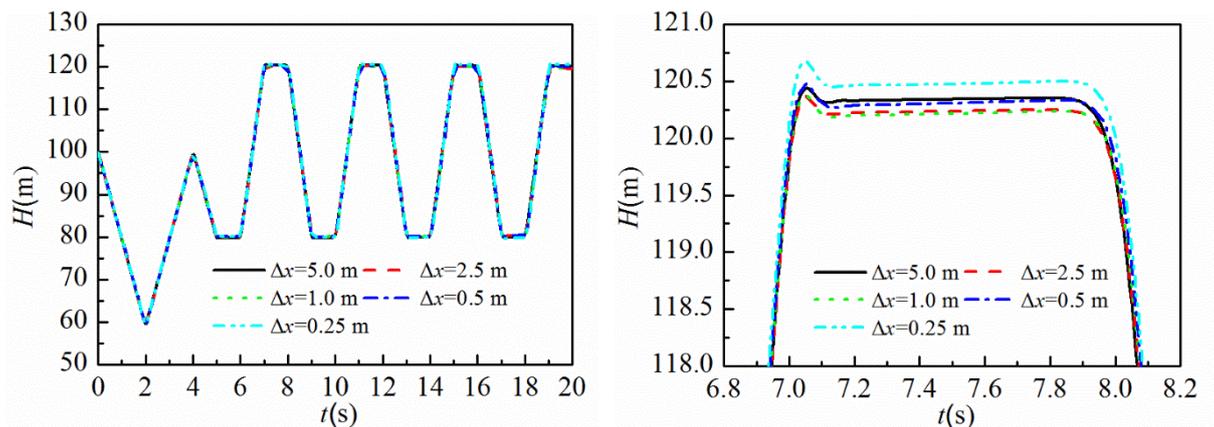


Figure 14: Effect of 3D mesh size on coupling result. Left: overview, right: zoom-up of first peak

5.6 APPLICATION TO THE TEST RIG

The main purpose of the coupling method is to use it in complex hydraulic systems where a large part of the system can be modeled with 1D MOC and a small part of

the system must be modeled by 3D CFD. Here it is applied to the test rig. The results are compared with the experimental data.

The coupled method is used to model the pressure transient in the system. The gate region, including 1 m pipe upstream and 2 m pipe downstream, is modeled by CFD using a dynamic mesh (in 2D for time saving purpose). The upstream parts of the rig are simulated by 1D MOC, while the downstream tank is approximated using a constant outlet pressure of the CFD region since the sonicLiquidFoam solver cannot model free surfaces. Figure 16 shows that the calculated pressure head at the coupling interface 1 m upstream the gate is similar as the experimental results at that location. The pressure oscillations when the gate is closed are due to water hammer. The slight increase in the mean value during the oscillations is due to the increase in the upstream water level, which is included in the 1D approach. The simulated pressure head is higher than the experimental value when the gate is fully opened (at 20 s). This can be explained by the constant pressure boundary condition that is applied at the outlet of the CFD region, while the experimental water surface in the tank was lower than initially at that time.

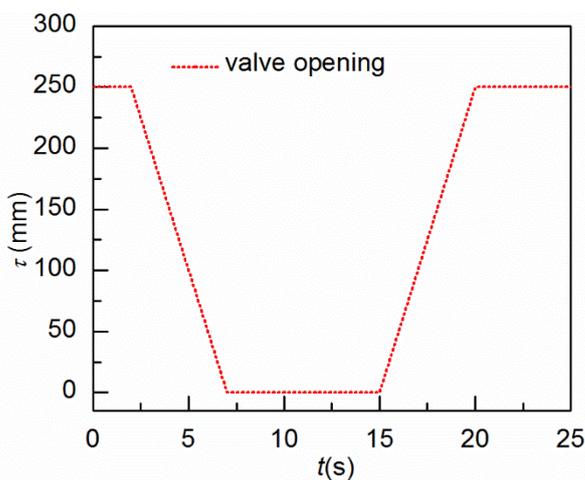


Figure 15: Gate position over time

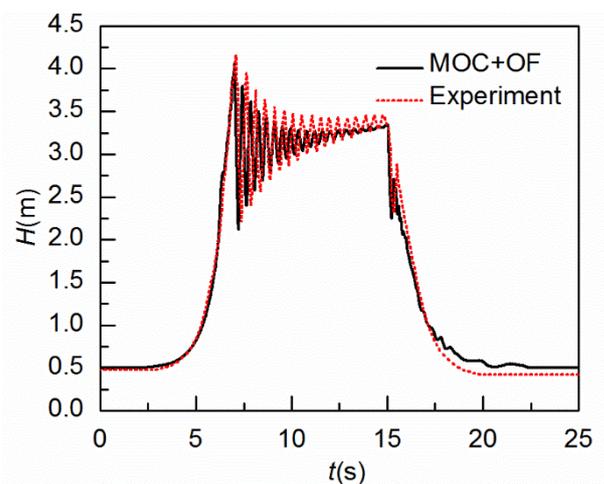


Figure 16: Coupled pressure head compared with experiment

6 Conclusions

The case is studied numerically with three approaches. The result from a 3D incompressible approach with free surfaces captures the main behavior of the flow well except for the pressure fluctuations that correspond to a finite sound propagation speed. It shows that the pressure fluctuations are not due to the free surfaces, and suggests that they are solely due to compressible effects.

It is found that the outlet boundary condition that represents the characteristics of the closing gate in a 1D approach is important to accurately predict both the pressure rise and pressure fluctuations. The hypothesis of a low speed of sound in this work due to a large amount of air in the water and due to the pipe flexibility is investigated. The good agreement between the experimental and numerical results confirms that a speed of sound that corresponds to water with a volume fraction of air of 0.1% is appropriate in the current case.

Finally, a 1D-3D coupling method is implemented to take advantage of the benefits of both the 1D and 3D approaches. The 1D region is simulated using MOC and the 3D region is simulated using the finite volume method. The accuracy and robustness of the coupling are verified for the case of water hammer in a straight pipe. It is shown that the time discretization scheme may influence the sharpness or the pressure pulsations, although the main properties are resolved well by all schemes. The coupling is finally applied to the test rig, and the results are similar as both the experimental data and 1D MOC results where the gate is included using its numerically determined static characteristics.

7 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., Chichester, 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.
- Petit, O., "1D system transients coupled with 3D local flow details", Energiforsk rapport 2015:109.
- O. Petit O., Wang C., Cervantes M. and Nilsson H., "Numerical and Experiment Investigations of a Hydraulic Channel During a Gate Closure at a High Reynolds Number", 6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problem in Hydraulic Machinery and system, September 9-11, 2015, Ljubljana, Slovenia.
- Wang C., Nilsson H., Yang J., Petit O., "1D-3D coupling for hydraulic system transient simulations", submitted for journal publication in Elsevier Journal Computer Physics Communication, 2015.
- Wylie, E. B., and V. L. Streeter, "Fluid transients", Textbook, Feb Press, Ann Arbor, MI, USA, 1982.

1D SYSTEM TRANSIENTS COUPLED WITH 3D LOCAL FLOW DETAILS

Här redovisas en numerisk studie av vattenströmningen i en fyrkantig sluten kanal i samband med att en ventil stängs. Vattenströmningen drivs av nivåskillnaden mellan de två vattenytorna i en tank uppströms och en tank nedströms. Målet är att efterlikna strömningen i vattenkrafttunnlar när ett aggregat plötsligt stängs. Det är då viktigt att kunna kvantifiera tryckvågornas storlek och frekvens i systemet.

Studien har tre olika angreppssätt – tredimensionella inkompressibla beräkningar av strömningar i hela systemet, endimensionella kompressibla beräkningar av tryckförändringen i kanalen respektive implementering och validering av en 1D-3D-kopplingsmetod som kombinerar de två första metoderna.

Energiforsk is the Swedish Energy Research Centre – an industrially owned body dedicated to meeting the common energy challenges faced by industries, authorities and society. Our vision is to be hub of Swedish energy research and our mission is to make the world of energy smarter!