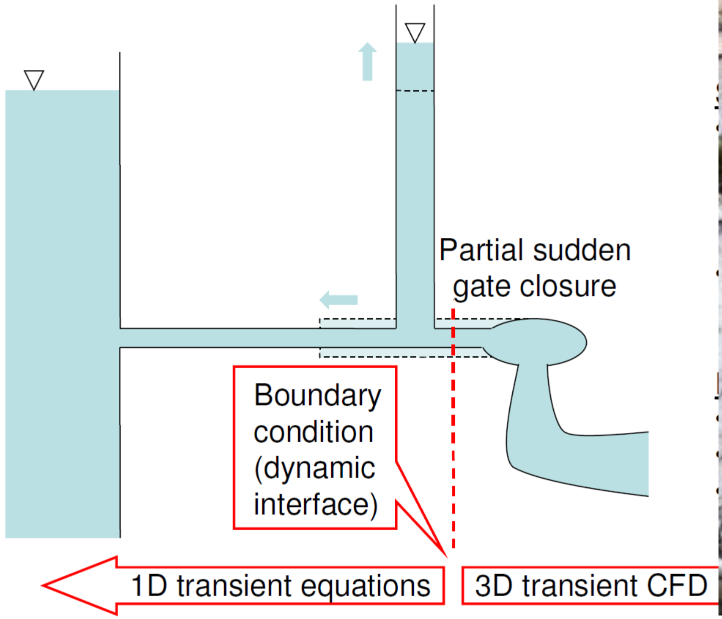
Numerical and Experimental Investigations of a Hydraulic Pipe during a Gate Closure at a High Reynolds Number

Olivier Petit

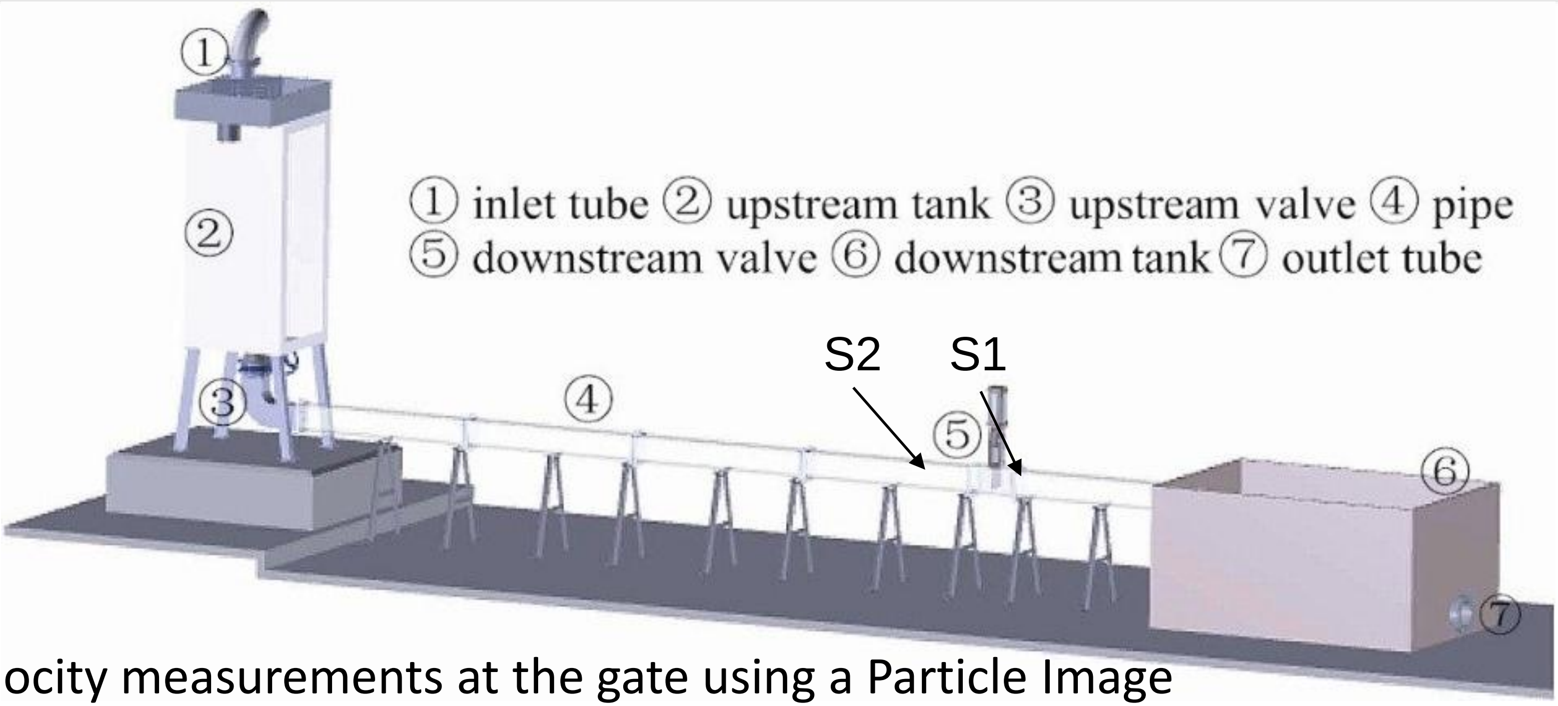
Objectives

- Study 1D-3D coupling and system transients coupled to local flow unsteadiness, such as start-stop, surge, water hammer, transient loads.

- Increase the computation speed by using 1D computation for easy geometry/flow feature (ex: penstocks) while required accuracy.



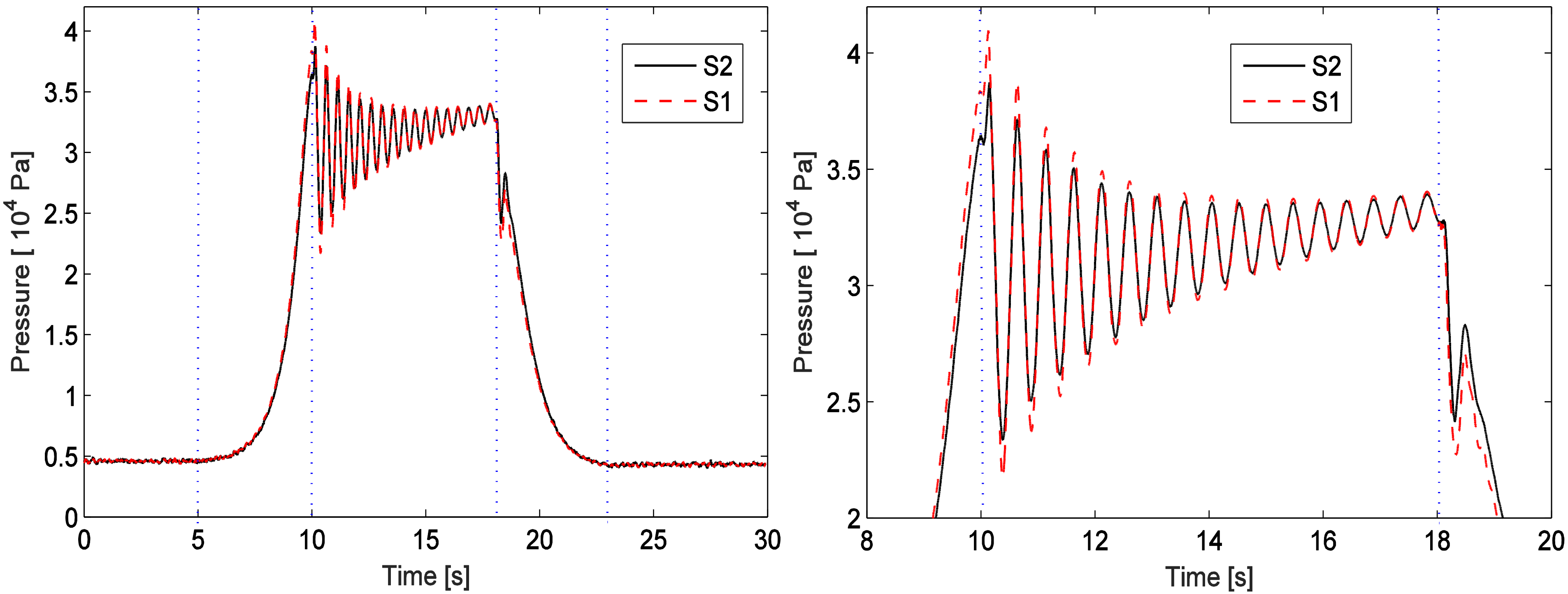
The test rig



- Velocity measurements at the gate using a Particle Image Velocimetry system (PIV).
- Time-resolved measurements of the static pressure along the channel: S1: 1.245 m upstream the gate, S2: 3.245 m upstream the gate.
- Constant volume flow during the experiments, Q=50l/s.

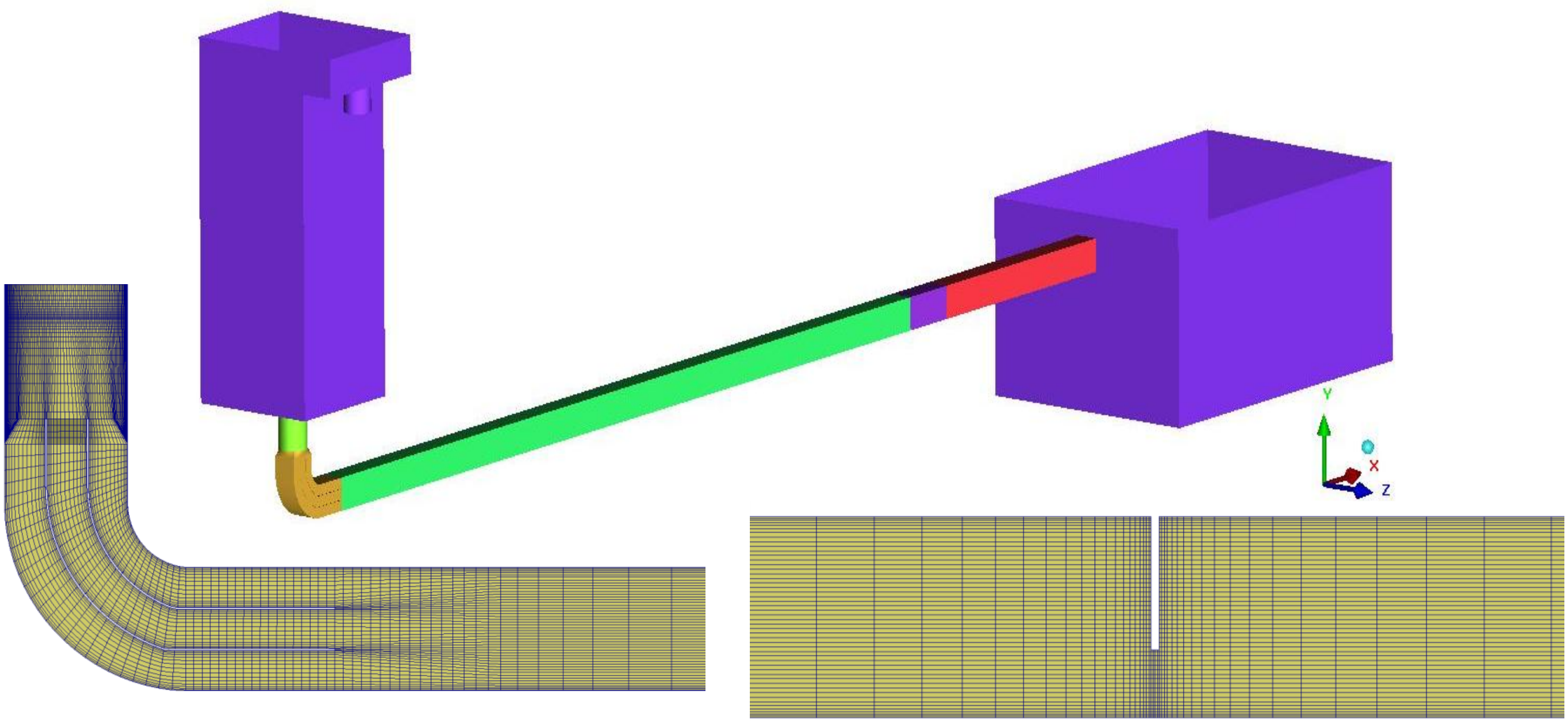
Measurements

- Oscillations in the pressure are present after complete closure with a period of about 0.48s.
- Water hammer effect (but very large oscillation period) or standing waves in the upper tank?

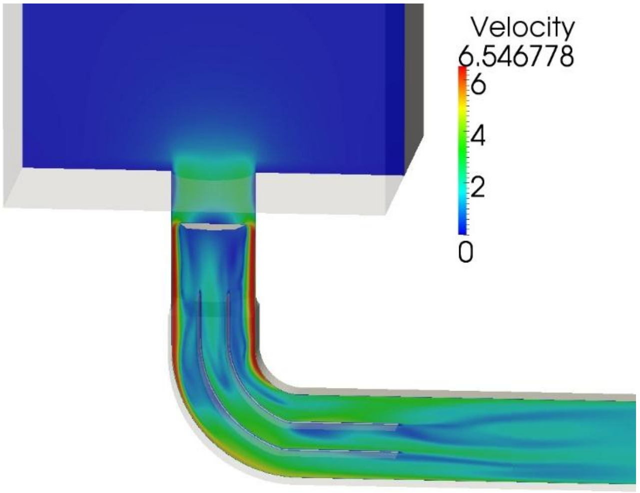


3D numerical investigation

- 2 millions nodes.
- General Grid interface between the two tanks and the straight pipe.
- Free surface at the upstream and downstream tanks.

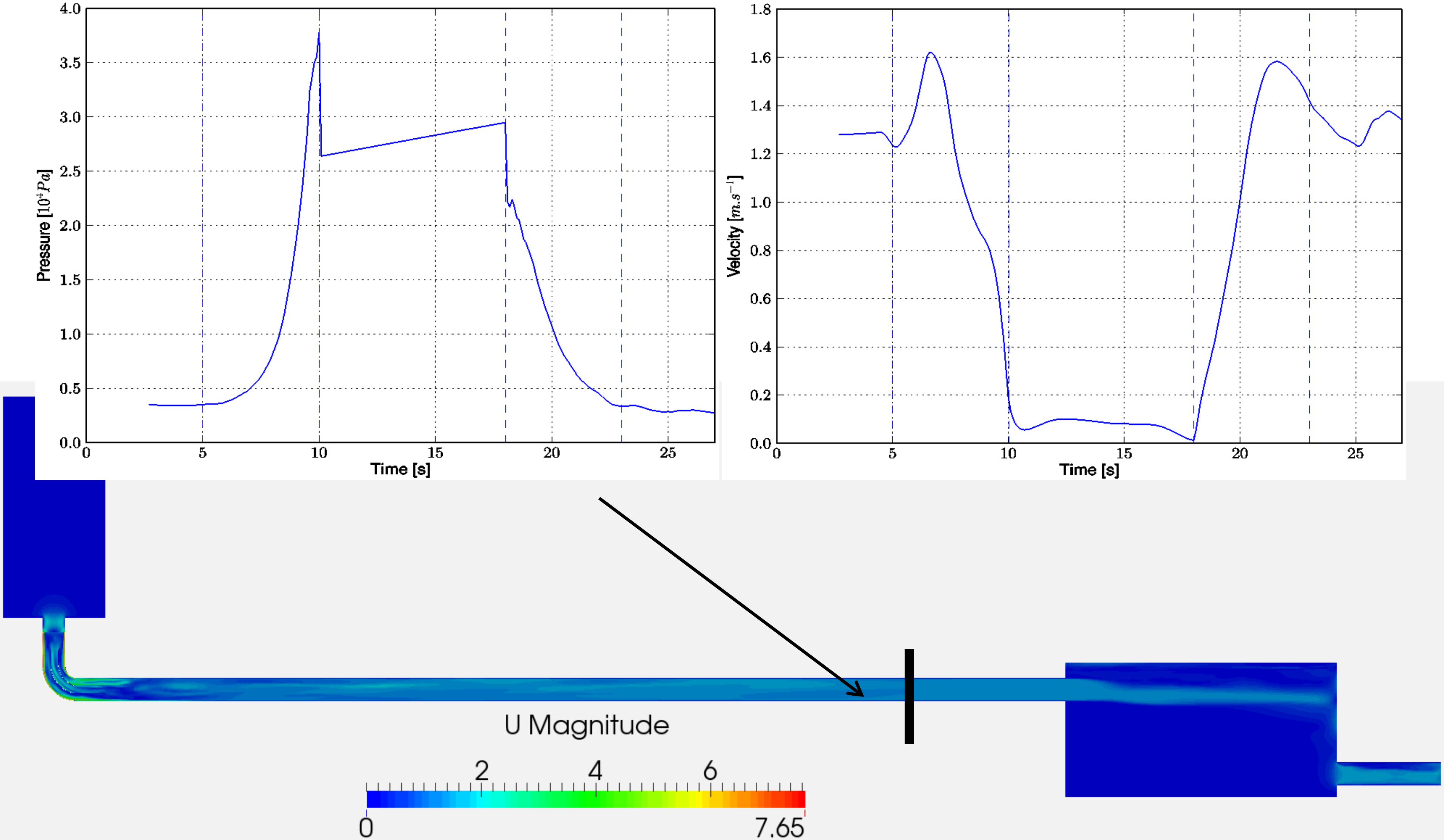


3D numerical investigation



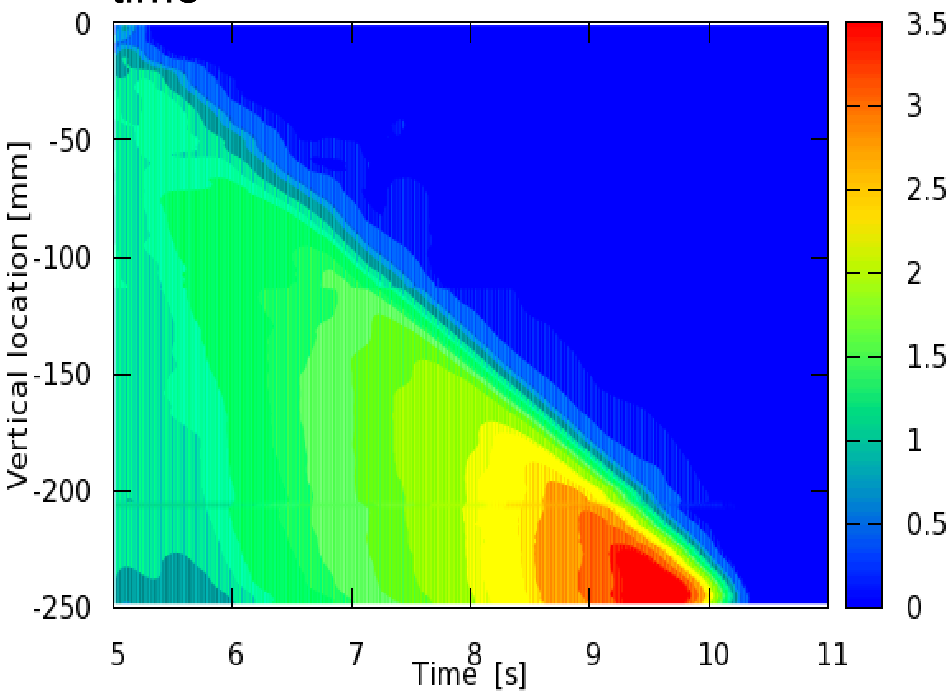
- FOAM-extend 3.1.
- No turbulence model.
- Constant volume flow Q=0.05 m3/s.
- The interDyMFoam solver used: incompressible two-phase flow solver combine with Dynamic Mesh.
- The transient starts at 5 s, after a stabilization period, followed by 4.9 s closing, 8 s closed and 4.9 s opening gate.
- The 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.

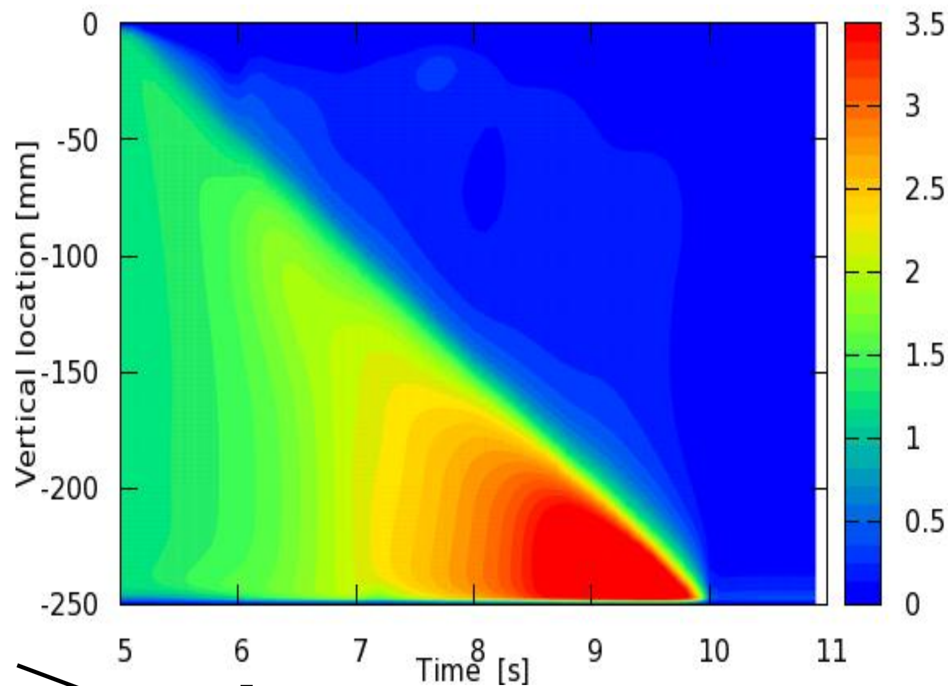
3D numerical investigation

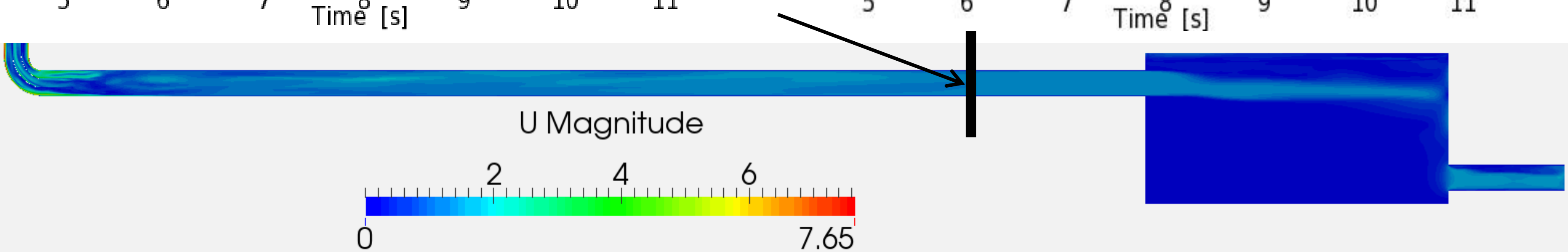


3D numerical investigation

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

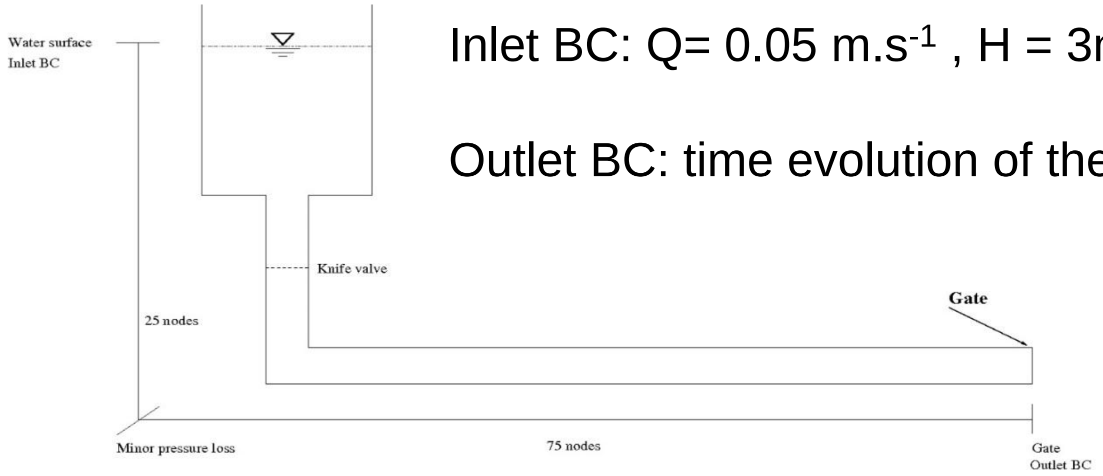






1D numerical investigation

- 1D numerical investigation using Method of Characteristics (MOC).
- A reservoir-pipeline-gate system is used to compute the 1D behavior of the flow.
- The grid size is 0.08 m, based on the sound speed and the time step to keep the Courant number at one.
- The air contained in the water as well as the structure flexibility lower the air speed. In this investigation, the air speed is chosen as a = 80 m.s-1.

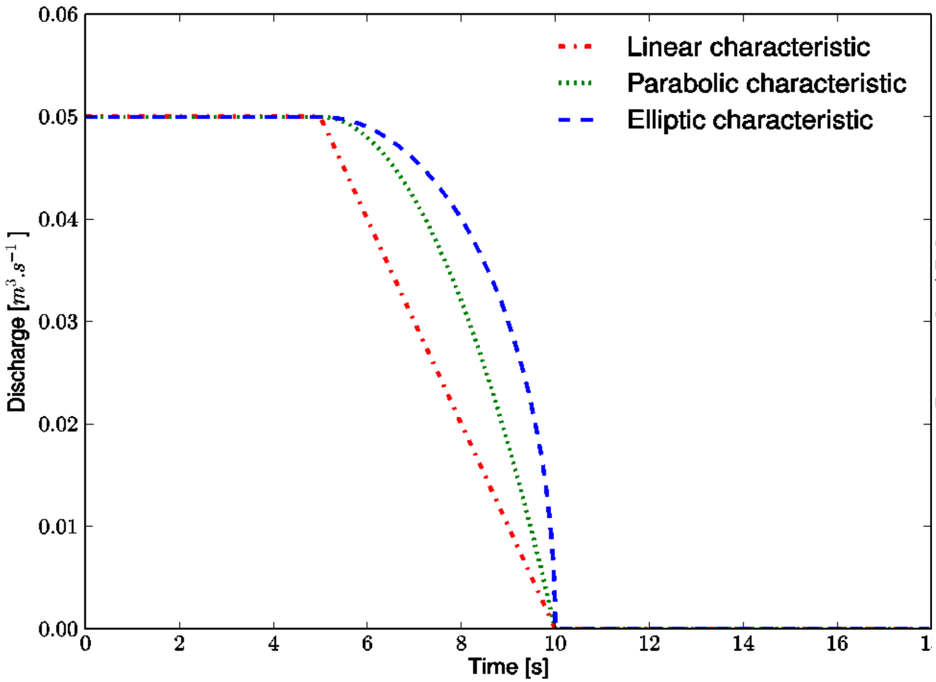


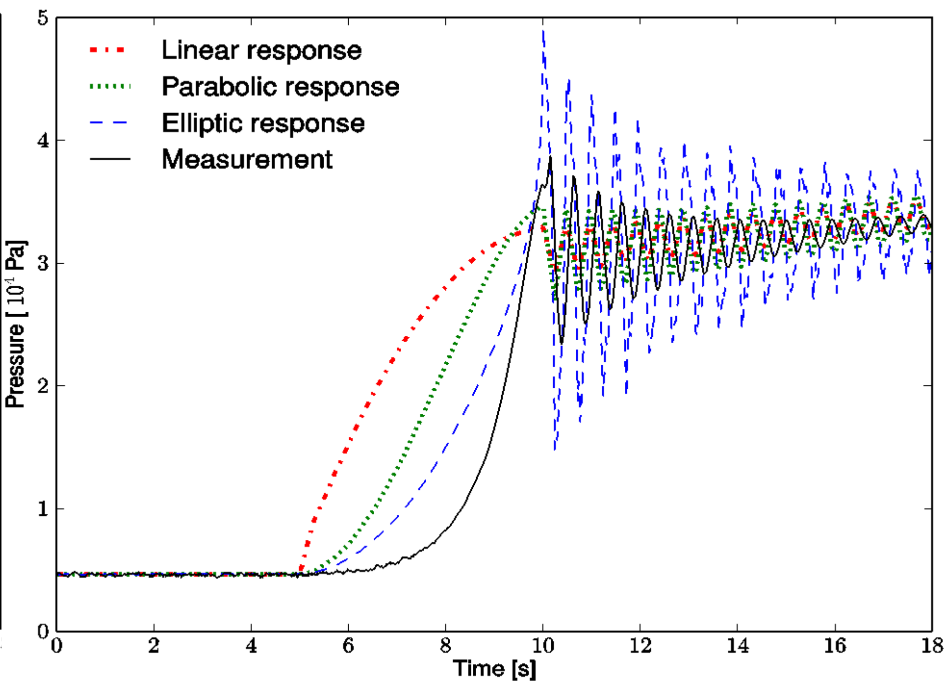
Inlet BC: Q= 0.05 $m.s^{-1}$, H = 3m.

Outlet BC: time evolution of the discharge.

1D numerical investigation

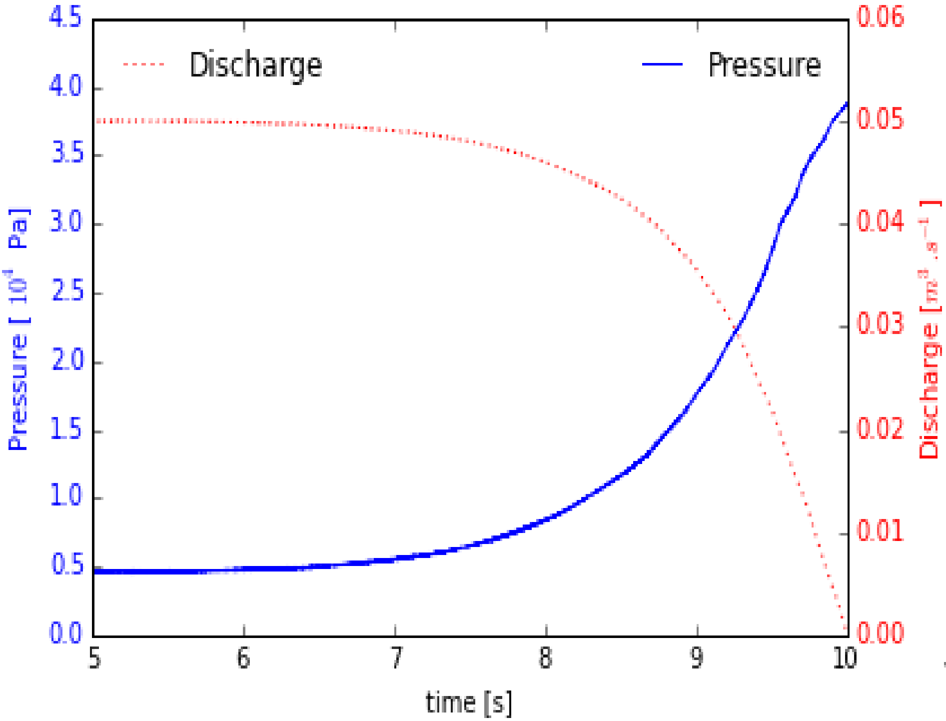
Outlet BC: importance of the time evolution of the discharge





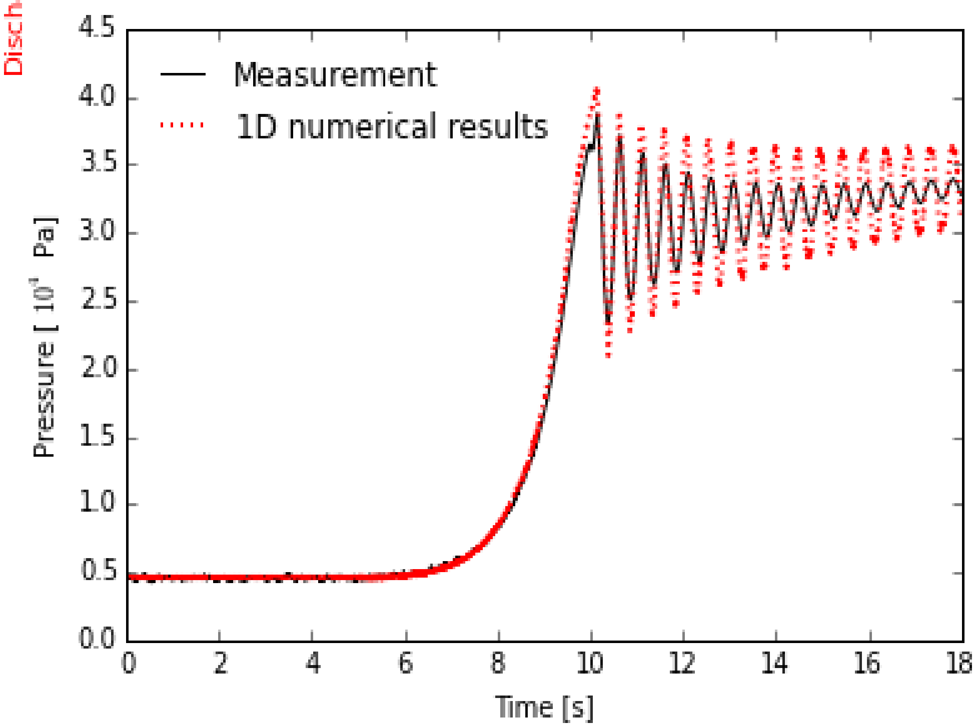
Approximate gate closing characteristic (left) and computed pressure response compared with the measurements (right).

1D numerical investigation



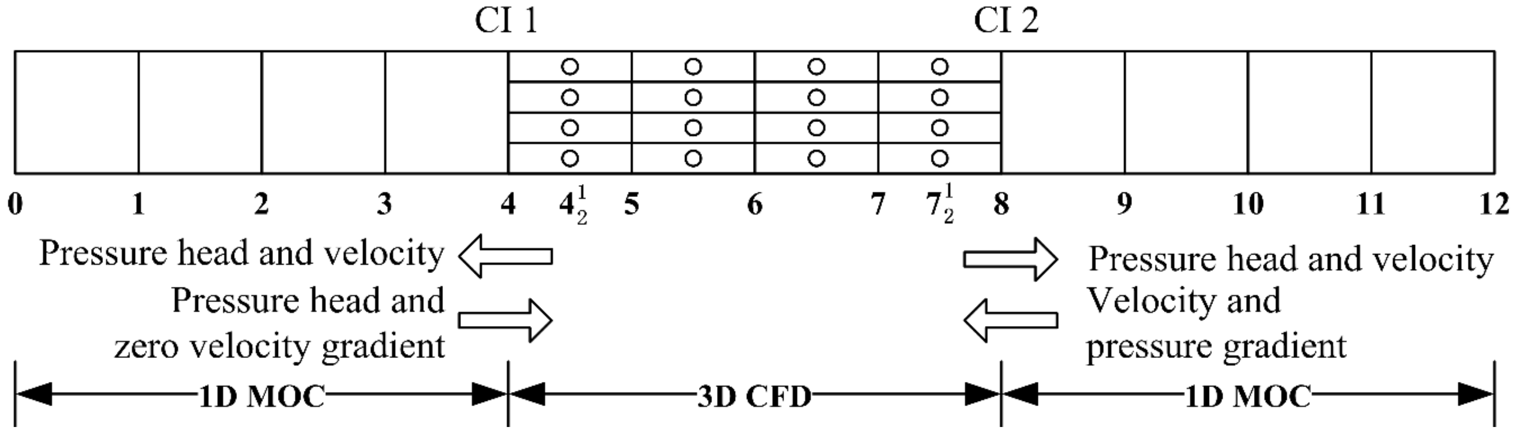
Measured pressure and corresponding approximated discharge, as the gate closes.

The pressure waves are not damped as much as in the experiment. This is probably due to a lack of a dynamic pressure loss term in the 1D MOC model.



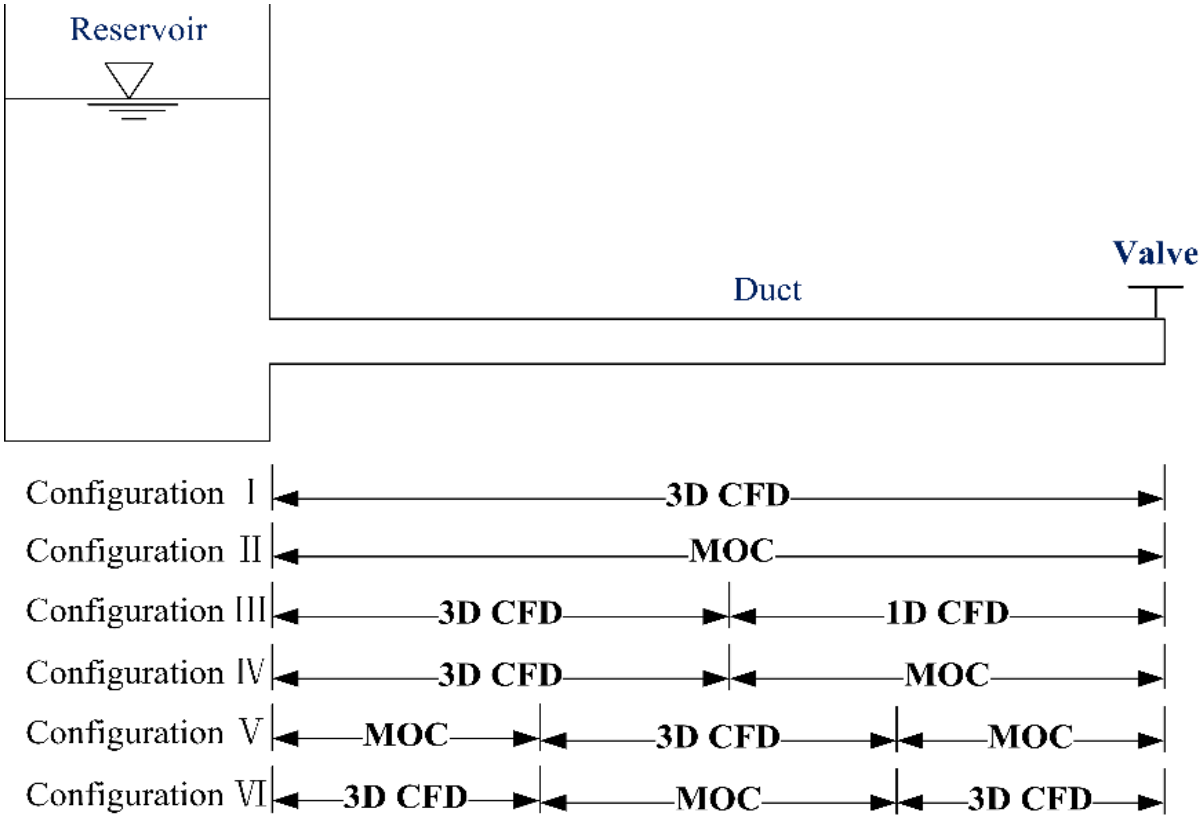
1D-3D coupling

- 1D modeled using the MOC implementation
- 3D computed with sondicLiquidFoam
- 1D-3D coupling based on Riemann invariants



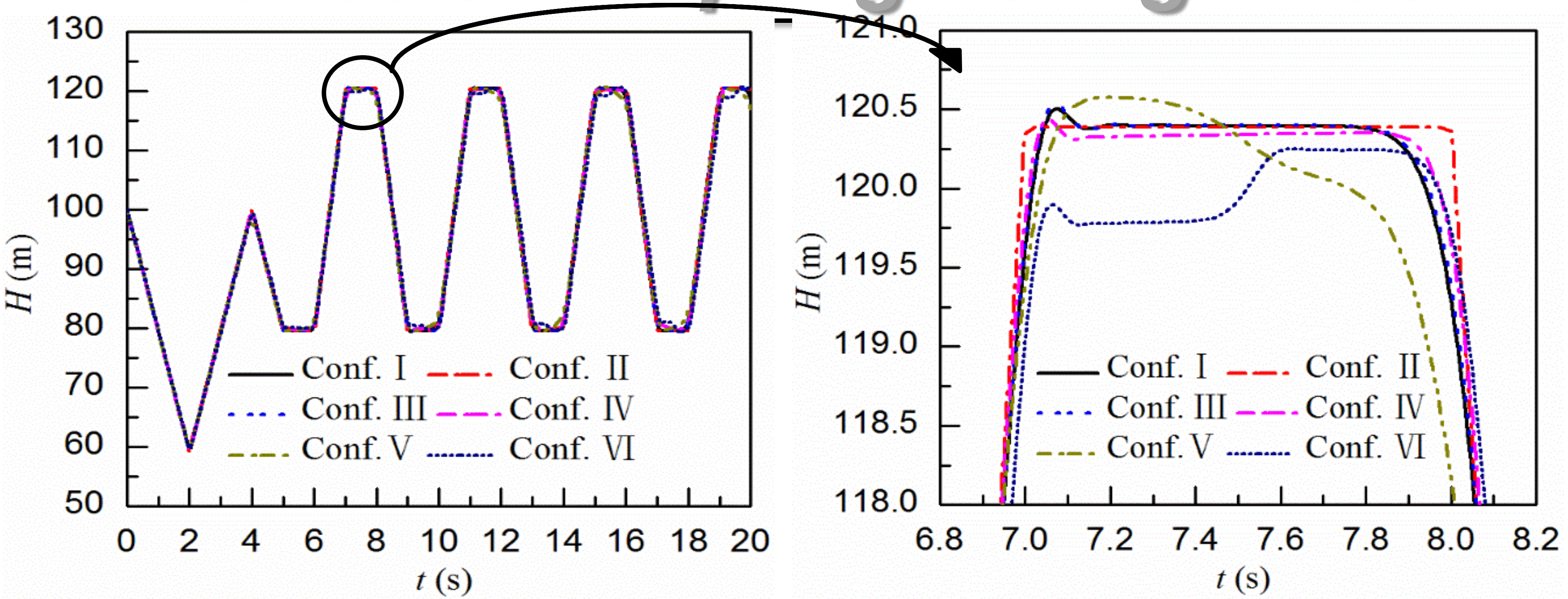
MOC-CFD-MOC coupling configuration for a duct

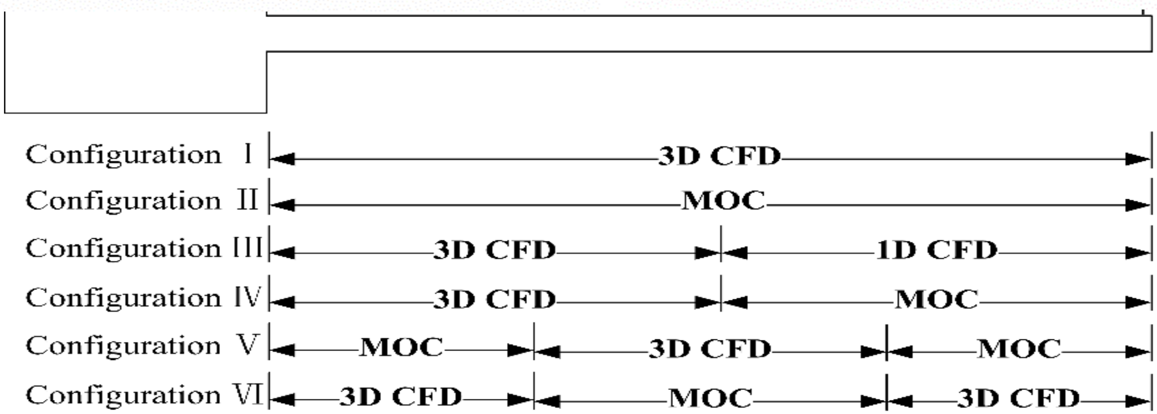
1D-3D coupling – validation case



- 1000 m long duct.
- 100 m head at the entrance of the duct.
- Linear increase of the duct outlet discharge from 0 to 1 m^3/s in 5 s.
- $\Delta t = 0.005$ s
- Mesh size of 5 m along the duct for both models.
- Conf. III uses a GGI interface between the two regions.

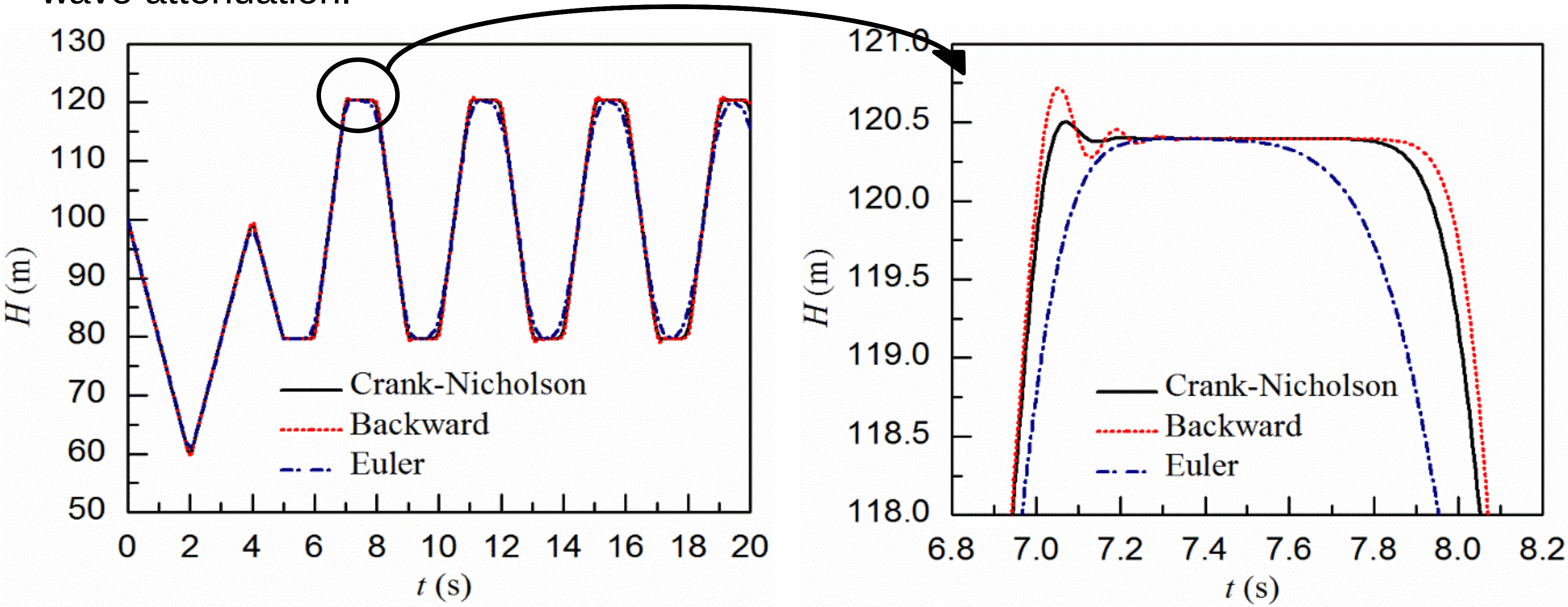
Influence of coupling configuration





Discretization scheme

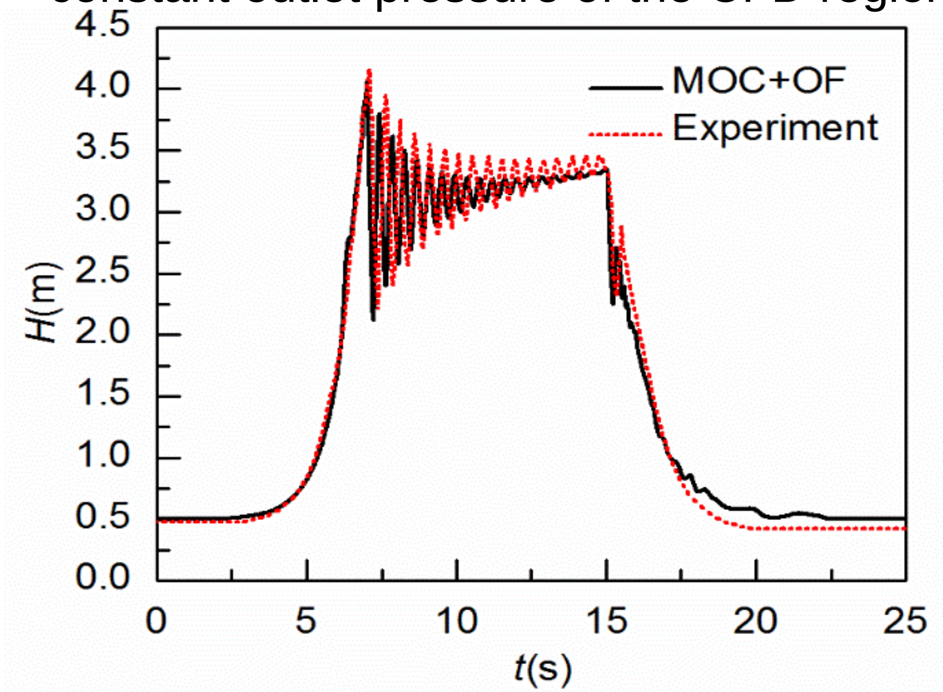
Aim: investigate the influence of the time discretization scheme on the pressure wave attenuation.

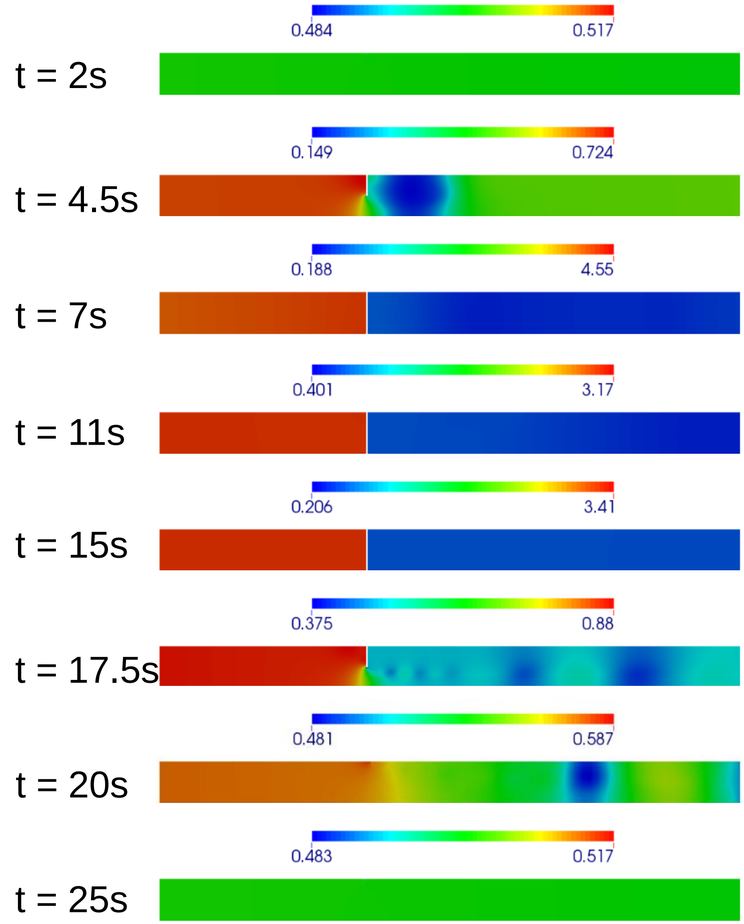


- Second order Crank-Nicholson scheme is used with a blend factor of 0.45.
- The chosen configuration for this investigation is Configuration I (pure 3D CFD)

1D-3D coupling – back to the test case

- The gate region, including 1 m pipe upstream and 2 m pipe downstream, is modeled by CFD using a dynamic mesh in 2D.
- The solver is sonicLiquidFoam.
- The upstream parts of the rig are simulated by 1D MOC
- The downstream tank is approximated using a constant outlet pressure of the CFD region.





Conclusions

- An experimental rig was designed. Time-resolved static pressure measurements and PIV measurements form experimental database.
- 3D numerical investigation shows that the pressure fluctuations are not due to the free surfaces, and suggests that they are solely due to compressible effects.
- 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 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.
- The 1D-3D coupling implemented in OpenFOAM seems to accurately predict the pressure rise and fluctuations due to water hammer.

Thank you for your attention!