Transactions, **SMiRT-26**
Berlin/Potsdam, Germany, July 10-15, 2022
Division III

BEST-ESTIMATE WATER HAMMER SIMULATIONS TO AVOID THE CALCULATION OF UNREALISTICALLY HIGH LOADS OR UNPHYSICAL PRESSURE AND FORCE PEAKS

Thorsten Neuhaus[1]

[1] Technical Expert, TÜV NORD EnSys, Hamburg, Germany (tneuhaus@tuev-nord.de)

ABSTRACT

For safety-related systems fluid loads due to fluid transients have to be quantified for subsequent structural analyses to ensure their integrity or function, as required. Usually transient fluid loads in pipe systems are determined with one-dimensional water hammer software. For single-phase liquid flow, the method of characteristics (MOC) is often used that gives in this case appropriate results. For the consideration of local vapor bubbles, the MOC is combined with the discrete vapor cavity model (DVC). The DVC model may generate unrealistic pressure spikes due to the calculation of the collapse of multi-cavities in scenarios, where only one vapor bubble should actually occur. The application of a two-phase code may improve the calculation results. One requirement for the latter codes is the ability to calculate the propagation of steep gradients without suffering from numerical diffusion to exclude the underestimation of fluid loads. This is commonly attained by applying higher-order numerical schemes. However, the application of a numerical method of pure 2nd order leads to the calculation of unphysical oscillations at steep gradients causing severe problems during the solution. To exclude this, numerical methods with flux limiters can be used. With their application, the calculation of unrealistically high loads due to numerical deficiencies can be minimized. In addition, the consideration of further physical effects, that lead to the reduction of loads during transient flow processes, allows for a more realistic calculation of the loads. These are unsteady friction, widening of the pipe caused by pressure increase, fluid-structure interaction at junctions and due to friction, degassing of gas that is initially dissolved physically in a liquid and thermodynamic non-equilibrium during vapor bubble collapse. The in-house code DYVRO applies a second-order accurate scheme with flux limiters based on the Godunov method and can account for the above-described physical phenomena. It is shown by comparison of calculation results obtained by DYVRO with experimental data from literature that with modeling of these physical effects the loads can be calculated more realistically. Generally, these loads are lower than the results calculated by simplified models, which do not account for these effects. Considering that these loads are applied in subsequent structural analyses, cost-intensive oversizing of pipes and their supports can be avoided, by ensuring the necessary safety.

INTRODUCTION

In piping systems, the actuation of pumps and valves or the break of a pipe may result in large and rapid pressure fluctuations respectively water hammers, which generate loads on the pipes and their supports. For a safety-related system, these loads have to be quantified for subsequent structural analyses to ensure their integrity or function, as required. For example, the codes ASME B31.1 (2010), ASME B31.3 (2016) and ASME B31.4 (1998) specifically indicate that loads due to fluid transients shall be considered. Piping systems may be e.g. long-distance pipelines with lengths up to hundreds of kilometres or complex branched pipe systems like a water-steam system in a power plant that may consist of dozens of pumps, valves, heat exchangers etc. Hence, specialised one-dimensional water hammer software is commonly used for the computation of transient fluid loads in pipe systems.

MODELING APPROACHES FOR FLUID FLOW IN PIPE SYSTEMS

For the simulation of pressure surges in pipe systems usually a one-dimensional approach is applied. For single-phase liquid flow, for which an almost constant density and sound velocity can be assumed, the method of characteristics (MOC) is often used, described for example in Wylie and Streeter (1993). If the time step size and the spatial discretization are chosen in such a way that the conservation equations are solved along the characteristic lines, the propagation of sharp pressure peaks will be calculated by the MOC without undesired numerical effects like numerical diffusion, that is the smearing of steep gradients as they progress, or unphysical oscillations. However, since the time step applies to the entire calculation model, a uniform spatial discretisation is required for this. Despite its simplicity, the MOC is one of the most suitable methods for the calculation of transient, single-phase liquid flows in pipe systems, if it is possible to use a uniform spatial discretisation in the investigated pipe system. Pressure surge programs like Flowmaster (since 2016 FloMASTER), AFT Impulse, Wanda, Pipenet or Sir 3S apply the MOC.

For the calculation of local vapor bubbles, the MOC is often combined with the discrete vapor cavity model (DVC) that is widely used in standard water hammer software packages. The DVC model allows vapor cavities to form at computational sections in the MOC, when the pressure falls below the liquid vapor pressure. At these locations the classical water hammer solution is no longer valid. Instead, the pressure is set to vapor pressure and the section acts as a pressure boundary condition for the adjacent pipe sections. Two interfaces between liquid and vapor are assumed upstream and downstream of the cavity with a shape corresponding to the cross-sectional area of the pipe and perpendicular to the pipe axis. The movement of the interfaces is calculated from the fluid velocities in the adjacent pipes. When the phase interfaces reach each other, a collapse of the vapor cavity is calculated together with a sudden pressure rise. The vapor is not treated as a phase with specific material data, but only as a cavity as the name of the DVC model already says. The simulated vapor cavity cannot move away from the calculation node, where it was initially generated. As it is shown later in this paper the DVC model has some undesired properties, e.g. it may generate unrealistic pressure spikes due to the calculation of the collapse of multi-cavities in scenarios, where only one vapor bubble should actually occur, as pointed out in Bergant et al. (2007).

For gaseous single-phase or two-phase flow the material data of the fluid, like density and sound velocity, change in time and space. In these cases the equations for the conservation of momentum, mass and energy are usually solved using the Finite Element Method (FEM), the Finite Difference Method (FDM) or the Finite Volume Method (FVM). For two-phase flow the conservation equations are sometimes applied for each phase individually and sometimes for the mixture. The one-dimensional, transient flow model consists mostly of a hyperbolic system of partial differential equations. Explicit numerical methods of 1st or 2nd order or combinations of these are usually used to solve these equations. Toro (2009) gives a good overview of numerical methods to handle hyperbolic partial differential equations. Explicit 1st order numerical methods, which are first order accurate in time and first order accurate in space, like the Lax-Friedrichs method or a standard 1st order upwind scheme, may be subject to numerical diffusion, which can lead to pressure gradients being smeared and thus fluid forces being underestimated. On the other hand, explicit 2nd order methods, such as the Lax-Wendroff or MacCormack method, can maintain steep gradients as they progress. However, undesirable unphysical oscillations can occur, when they are applied for solving the conservation equations. These oscillations can lead to serious problems during the computation,

- if the pressure level in the pipe system under investigation is low and negative pressures are calculated as a result of the oscillations or
- if in case of two-phase flow oscillations occur close to the saturation pressure causing the pressure to fall below the saturation pressure repeatedly, which will lead to the formation of vapor bubbles in scenarios where no vapor bubbles should actually occur, or
- if in case of a void fraction near 0 or 1 in a two-phase flow regime, as a result of the numerical oscillations a value for the void fraction of smaller than 0 or higher than 1 is calculated.

Therefore, unphysical, numerically caused oscillations should be excluded as far as possible to avoid the calculation of unrealistically high loads or unphysical pressure and associated force peaks, and to get a solution at all as well.

However, there is only a little number of commercial one-dimensional two-phase codes available like e.g. DRAKO, so that users sometimes apply system codes for load calculations like RELAP5 or ATHLET, which originally are not designed for this task. For example, they cannot account for the widening of the pipe caused by pressure increase, the so-called Poisson coupling. Additionally, care must be taken regarding the numerical parameters like the nodalization and the time step, so that numerical diffusion or unphysical overshoots will be of minor relevance. Furthermore, the sub-models of some components, which are relevant for the simulation of the fast transients, like e.g. swing or lift check valves, may not be suitable for the simulation of water hammers.

Water hammer software like the WAHA-Code (see Gale and Tiselj, 2004) or DYVRO mod. 3 (see Neuhaus, 2009) solve the conservation equations for mass, momentum and energy of the fluid with a modern second-order accurate scheme with flux limiters, based on the Godunov method, that is appropriate for shock-capturing without unphysical oscillations. A description can be found in Toro (2009). This scheme also allows for different discretizations in various pipe segments of a pipe system. This may be necessary, when a complex pipe system consists of several small and large pipe sections with varying lengths. In such situations the application of the MOC could be disadvantageous, because without a uniform spatial discretization there will be undesired effects due to numerical diffusion.

FURTHER PHYSICAL EFFECTS AND THEIR MODELING

Several physical effects may be present during water hammer events, which may lower the loads on the pipe system, e.g. unsteady friction, widening of the pipe caused by pressure increase (Poisson coupling), fluid-structure interaction (FSI) at junctions and due to pipe wall friction, degassing of gas that is initially dissolved physically in a liquid as well as thermodynamic non-equilibrium during vapor bubble collapse. The detailed mathematical modeling of these effects is too extensive to be presented in this publication. Instead, reference is made to the relevant literature in the following.

a) Unsteady friction
The calculation approaches for friction forces at steady pipe flow underestimate the observed frictional effects during transient events. To account for unsteady friction models are available that use additional partial derivatives of the fluid velocity with respect to space and/or time, e.g. shown Vítkovský (2006). With these models, the faster damping of pressure and velocity signals can be better reproduced than with quasi-steady friction approaches.

b) Widening of the pipe caused by pressure increase
Due to the flexibility of the pipe wall material the pipe diameter grows with increasing internal pressure. This in turn reduces the pressure rise compared to the case where the pipe is considered rigid. The simplest way to account for this effect is the use of a lower pressure wave propagation speed that can be determined before the actual pressure surge calculation. See, for example Wylie and Streeter (1993).

c) Fluid-structure interaction (FSI) at junctions and due to pipe wall friction
The fluid-structure interaction is caused by an exchange of momentum between the pipe structure and the fluid inside. Frictional forces along the pipe axis and pressure and momentum forces at junctions lead to the movement of the pipe system. This movement sometimes leads to higher maximum values of the pressure during a transient compared to the case where the pipe system is considered rigid. However, the pressure gradients and thus the fluid forces on the pipe segments are generally lower compared to the case without FSI. To capture the FSI effect, several fluid and structural programs have been coupled in the past. The coupling is generally realized by the simultaneous execution of the

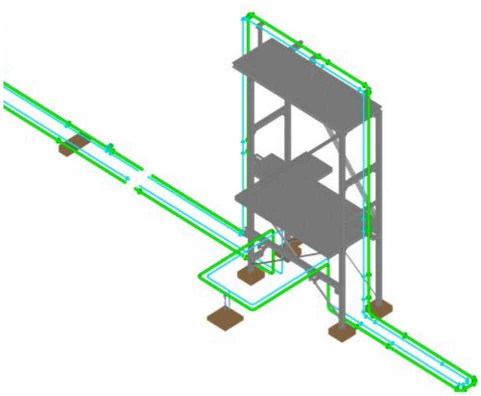
programs with exchange of accelerations and forces at relevant positions at each time-step. Details about FSI in pipe systems can be found in Tijsseling (1996) or Neuhaus (2019) for example.

d) Degassing of gas that is initially dissolved physically in a liquid
This effect may be relevant e.g. in tap water systems. The water is usually saturated with air that is dissolved physically. During pressure decrease, this air is released and induces damping effects during transient events. The degassing is not instantaneously but occurs with a time delay depending on the temperature of the water. In power plants commonly degassed water is used, so that this effect is not present except in systems carrying river or sea water. This effect is not modeled in most commercial water hammer software. One approach for the modeling of degassing is outlined in Neuhaus (2004).

e) Thermodynamic non-equilibrium during vapor bubble collapse
Especially in pipe systems at increased pressure and related high vapor density, the collapse of a vapor bubble and the associated pressure rise may be dependent on the speed of vapor condensation. With models that assume thermodynamic equilibrium between the liquid and vapor phase, i.e. same phase temperatures, the time-dependence cannot be captured well. Models accounting for non-equilibrium during vapor bubble collapse are more capable to reproduce the less steep pressure rise, that can be observed in practice during such processes, and the resulting lower fluid forces, please refer to Neuhaus (2011). Most commercially available water hammer codes do not consider this effect. However, system codes like ATHLET or RELAP5 are able to capture thermodynamic non-equilibrium.

DESCRIPTION OF THE TEST FACILITY

The set-up for the experiments used for the demonstration of the above-described effects in this paper, including the relevant measuring positions, is given in Figure 1. The experiments were conducted at the PPP test facility at Fraunhofer UMSICHT.



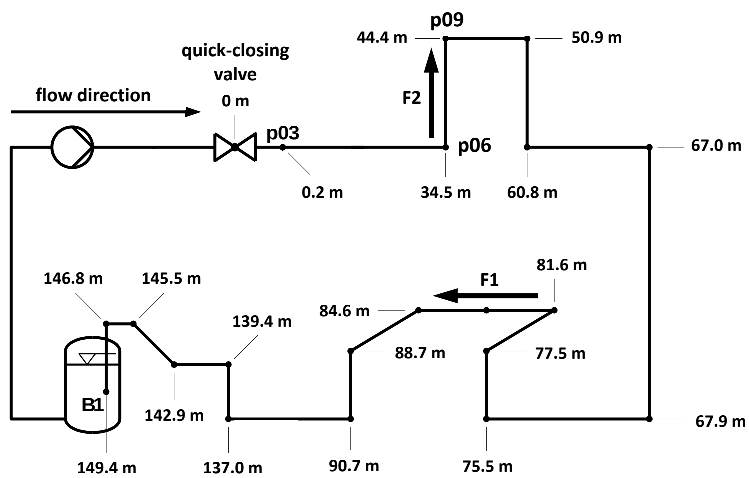


Figure 1. Scheme of experimental set-up and flow chart of the PPP test facility at Fraunhofer UMSICHT.

Demineralised tap water was pumped from the pressurised vessel B1 into the test system made of 108.3 mm inner diameter steel pipes with a wall thickness of 3 mm and having a total length of about 149.4 m downstream the quick-closing valve. All experiments started with a rapid valve closure. During the first phase of the transient process, a pressure wave traveled upstream the valve towards B1. On its way, it was partially reflected at the pump. Simultaneously a rarefaction wave traveled downstream the valve also towards B1. In the cases where the saturation pressure was reached, cavitation occurred directly downstream the valve and also in the upper part of the 9.9 m high pipe bridge, so that vapor bubbles were formed. The generated rarefaction and pressure waves oscillated in the pipe system until the cavities condensed, inducing cavitational water hammers. The processes upstream the valve are not considered in this paper. The pressure waves were measured by the fast pressure transducers P03, P06 and P09 with a frequency of 2 kHz. The fluid forces F1 and F2, whose location and direction are also included in Figure 1, are not determined directly with measurement technology, but are calculated from computed or measured pressures, as described in detail later in this publication. For more information about set-up and performance of experiments, please refer to Neuhaus (2005).

COMPARISON OF EXPERIMENTAL AND COMPUTATIONAL RESULTS

Three experiments are chosen for the comparison with computational results obtained with DYVRO:

- Exp. 415, initial pressure in B1: 19.65 bar(a), temperature: 21.9 °C, initial fluid velocity: 1.00 m/sec
- Exp. 132, initial pressure in B1: 1.14 bar(a), temperature: 20.3 °C, initial fluid velocity: 2.97 m/sec
- Exp. 307, initial pressure in B1: 9.92 bar(a), temperature: 119.7 °C, initial fluid velocity: 3.99 m/sec.

Experiment 415

Experiment 415 is the first used for validation, because vaporisation and air release processes did not occur. So the main focus can be set on the effects of widening of the pipe caused by pressure increase (Poisson coupling), unsteady friction as well as FSI at junctions and FSI pipe wall friction. In Figures 2 and 3, the measured and calculated pressures at P03 for experiment 415 are compared.

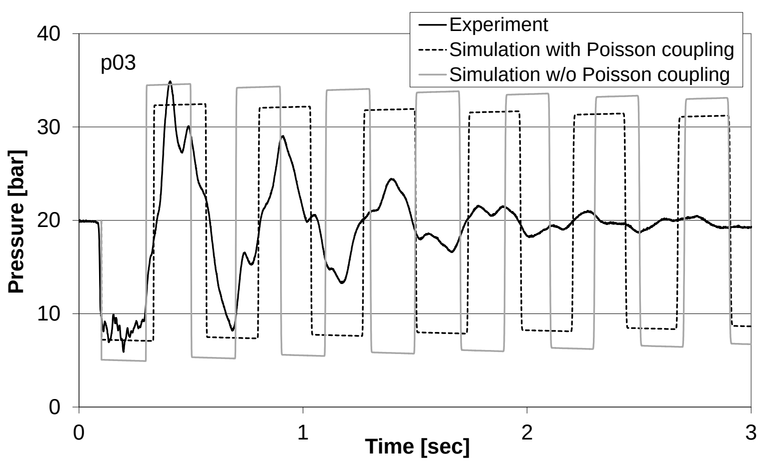


Figure 2. Measured and calculated pressure time histories downstream the valve for exp. 415 (1).

The calculation results in Figure 2 do not account for FSI, air release and unsteady friction. The shape of the measured pressure curve is not captured, but the model that considers the Poisson coupling reproduces the frequency of the pressure wave oscillations correctly. This is not the case with the model that does not consider the Poisson coupling. However, it is difficult to make a statement about the correctness of the pressure amplitudes in Figure 2 because secondary pressure waves due to FSI have a big influence on the maximum pressure values of the experimental data. Important in experiment 415 is the pressure damping with time that is captured well by the unsteady friction model shown in Figure 3. Here, the unsteady friction parameters are determined in adaption to the experimental setup. Nevertheless, the shape of the measured pressure curve cannot be reproduced, because it includes secondary high-frequency oscillations. Moreover, a more triangular shape of the experimental pressure curve appears in contrast to the rectangular shape of the simulated curves. Figure 3 shows that the introduction of the FSI-model strongly improves the simulation results. Thereby the maximum pressure at 0.45 sec can be reproduced.

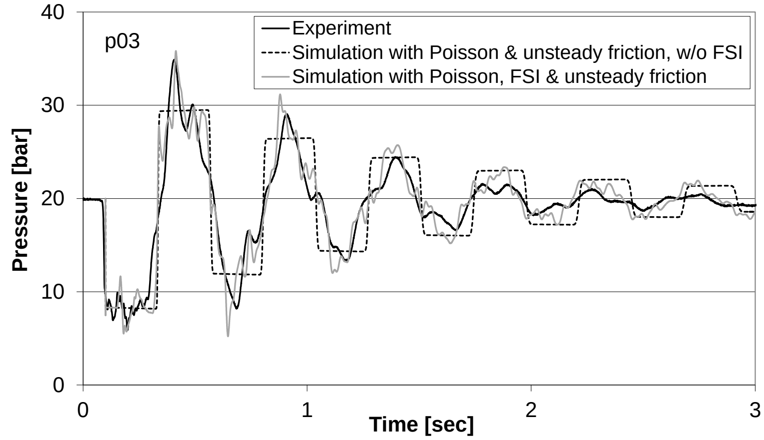


Figure 3. Measured and calculated pressure time histories downstream the valve for exp. 415 (2).

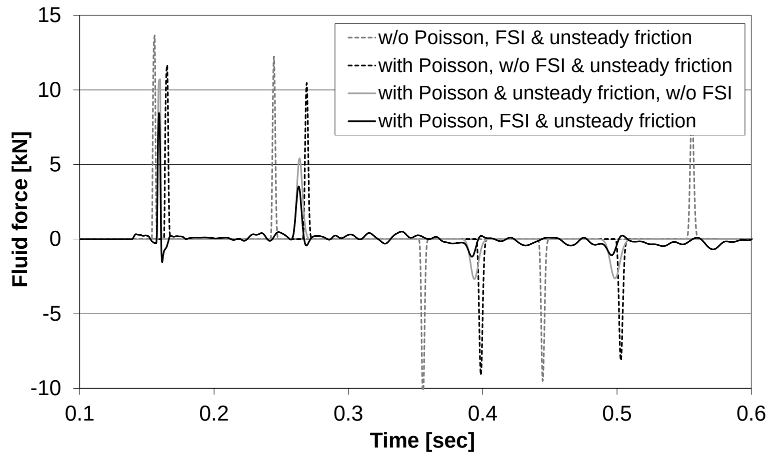


Figure 4. Calculated time histories of the fluid force F1 for exp. 415.

In Figure 4 the calculated time histories of the fluid force F1 (see Figure 1) by the four approaches are shown for the first 0.6 sec, in which the first rarefaction and pressure wave occur. It should be mentioned that the decrease of the pressure gradient due to the displacement of one single pipe segment may reduce the fluid forces on multiple pipe segments, because pressure waves travel through the pipe system. From Figure 4 it can be derived that the maximum calculated force by the most realistic model considering Poisson coupling, unsteady friction and FSI is 8.5 kN. Thus, it is significantly lower than the results of the other models (13.7 kN, 11.7 kN and 10.7 kN). The largest force in the opposite direction calculated by the most realistic model is 1.2 kN at t = 0.39 sec. Thus, it is significantly lower than the values that can be obtained with the models not considering unsteady friction, which are 9.1 kN respectively 10 kN.

Experiment 132

During experiment 132 large cavitational bubbles arose directly downstream the closed valve and in the upper part of the 9.9 m high pipe bridge. The vapor bubble in the pipe bridge collapsed for the first time after 14 sec, leading to a second pressure wave. In Figures 6 and 7 the measured and calculated pressures at P03 for experiment 132 are compared. In Figure 6 the results obtained by the MOC in combination with the DVC model are shown. It is apparent that the damping of the pressure waves in the experiment cannot be captured. Moreover, unrealistically sharp pressure spikes are calculated by MOC/DVC at 14.3 sec and

19 sec, which can result from the numerical effect of calculating the collapse of multi-cavities and which are associated with the calculation of high fluid forces.

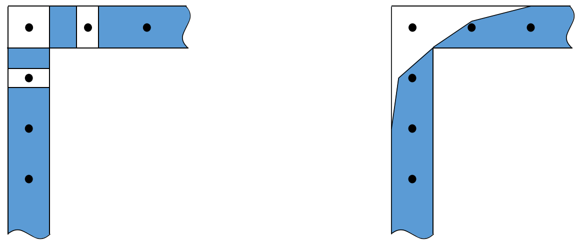


Figure 5. Modeling of a vapor region within a liquid: DVC model (left) and two-phase model (right).

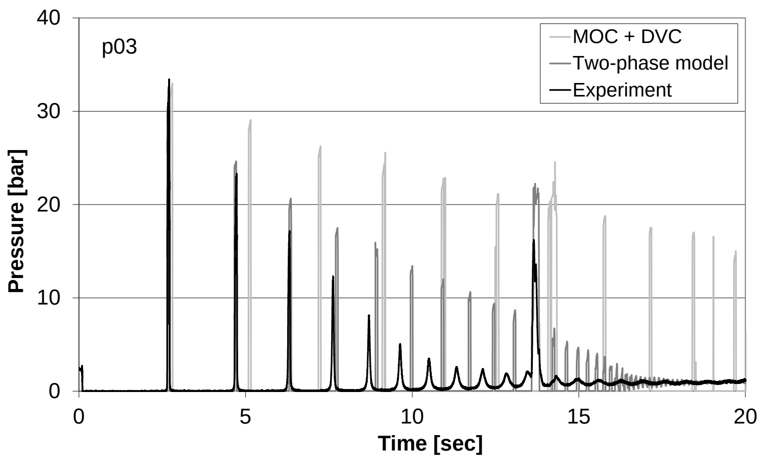


Figure 6. Measured and calculated pressure time histories downstream the valve for exp. 132 (1).

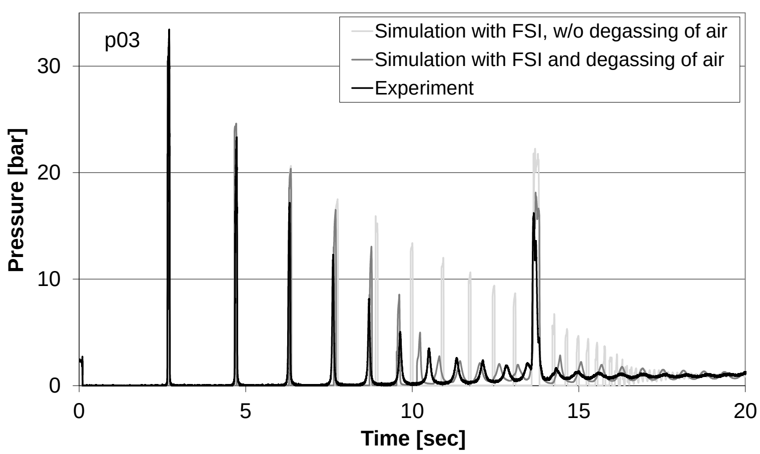


Figure 7. Measured and calculated pressure time histories downstream the valve for exp. 132 (2).

In Figure 6 also the results of a simple two-phase model are shown. The model does not include unsteady friction and air release, but FSI and vaporous cavitation. The damping and the frequency of the pressure surges after 7 sec cannot be reproduced by the model. However, numerical artefacts during the collapse of the vapor bubbles downstream the quick-closing valve and in the pipe bridge do not occur. The reason for this is the different treatment of vapor regions as shown in Figure 5. As described above, with a two-phase model a continuous extended vapor region, which can also move along the pipe axis, can be calculated, but not with MOC/DVC.

The simulated pressure at P03 using the air release model with the quasi-steady friction approach is shown in Figure 7. The frequency of the pressure oscillations at 14 sec can be captured well because the wave propagation speed, which drops due to the increasing compressibility of the water-air-mixture to about 250 m/sec, is calculated correctly. After 10 sec the shape of the experimental and simulated pressure curves look like a row of several ‘U’s. This effect can be reproduced by the model due to the dependency of the wave speed on the void fraction and the gas/vapor density. The air release leads also to higher damping of the pressure waves.

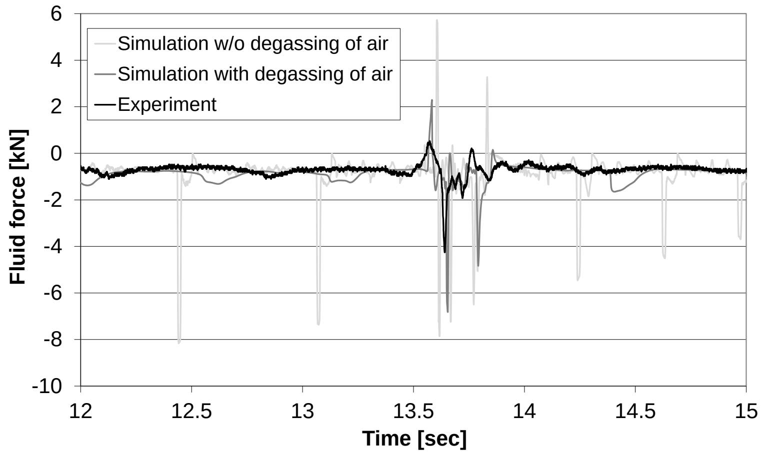


Figure 8. Time histories of the fluid force F2 for exp. 132.

The fluid force on a pipe segment can be calculated in a simplified manner using the upstream and downstream pressure at this segment. This concept is sometimes referred to as the “Endpoint Pressure Method” in literature and is valid if the force due to the pressure difference is high in comparison to momentum and frictional forces, which is the case in the experiments considered here. From the pressure records at the pressure transducers P06 and P09 it is possible to calculate the fluid force F2 on the first 9.9 m long vertical segment of the pipe bridge (see Figure 1). The force determined in this way is labelled “Experiment” in the legends of Figures 8 and 10. In Figure 8, the calculated and “measured” time histories of the fluid force F2 are shown in the time range between 12 and 15 sec. The calculated minima and maxima of -8.2 kN and -6.9 kN respectively 5.7 kN and 2.3 kN are still lower and higher, respectively, than the measured ones of -4.3 kN respectively 0.5 kN. With the MOC in combination with the DVC model typical rectangular-shaped curves of pressure are calculated leading to a minimum calculated value for the force F2 of -19 kN at the point in time, when the vapor bubble on the pipe bridge collapses. So, the forces calculated by more realistic models are considerably lower than those calculated by simplified models, but they are still significantly higher than the ones occurring in the real test facility and therefore covering and suited for structural analysis.

Experiment 307

The previous experiments 415 and 132 were conducted at an ambient temperature of around 20 °C. Experiment 307 is performed at a much higher temperature of about 120 °C. When vapor bubbles are generated this results in a 65 times higher steam density, which in turn means that at the subsequent vapor bubble collapse more steam must condense compared to the case at a temperature of 20 °C. The rapidity of condensation affects the steepness of the pressure gradients and thus the fluid forces. The higher the system pressure, the higher the vapor density and the longer the condensation takes, which leads to a rather smooth pressure rise during a vapor bubble collapse as can be seen within the circle in Figure 9. There the measured and calculated pressures at P03 for experiment 307 are compared. The two applied models do not include unsteady friction and air release, but FSI and vaporous cavitation.

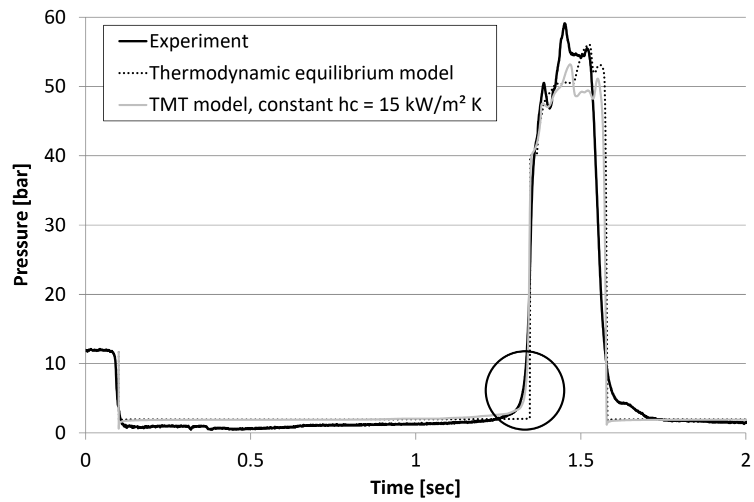


Figure 9. Measured and calculated pressure time histories downstream the valve for exp. 307.

The measured, rather smooth pressure rise at 1.35 sec can be better reproduced by the TMT model, that is a model accounting for thermodynamic non-equilibrium presented in Neuhaus (2011), compared to an equilibrium model assuming equal phase temperatures. In Figure 10 the time histories of the fluid force F2 are shown in the time range between 1.2 and 1.7 sec. The fluid force is not only dependent on the pressure amplitude, but also on the speed of the pressure increase, i.e. the pressure gradient. Therefore, the maximum absolute force value calculated with the TMT model is more than 30 % lower compared to the one calculated by a thermodynamic equilibrium model. But it is still significantly higher than the experimental data and therefore covering.

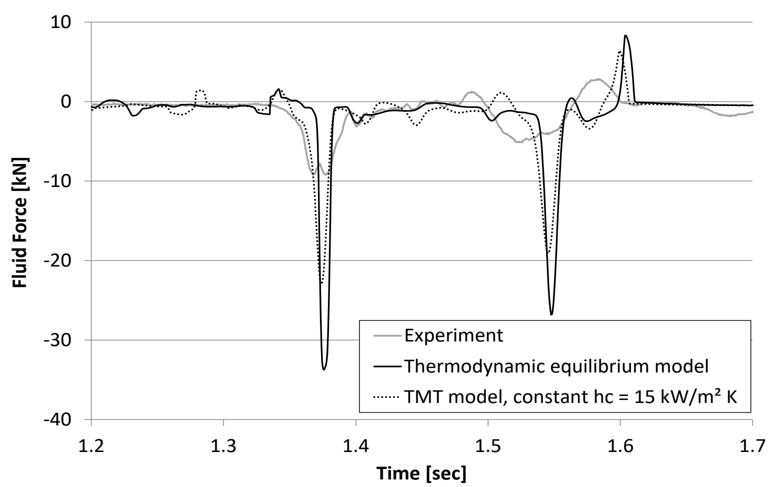


Figure 10. Time histories of the fluid force F2 for exp. 307.

CONCLUSION

It is shown by comparison with experimental data from literature that with a two-phase flow model that accounts for physical effects which can reduce the loads at pressure transients, the fluid loads acting on a pipe system can be calculated more realistically in comparison to simplified models. In addition, it is revealed that taking into account the effect of pipe wall widening at pressure increase is relevant for the correct calculation of the pressure wave frequencies and amplitudes. Since system codes like ATHLET and RELAP5 do not account for this effect, they should not be the first choice for the calculation of loads during rapid fluid transients. Moreover, it is demonstrated that in cases where multiple and larger vapor regions occur, the application of the MOC with the DVC model can lead to inappropriate results leading to too high forces. The application of a best-estimate code like DYVRO with a modern second-order accurate numerical scheme with flux limiters can avoid numerical artefacts like unphysical peaks or overshoots. Moreover, such an approach is suited to obviate the computation of unrealistically high fluid loads, since it is possible to consider relevant physical effects such as FSI or degassing of dissolved gases. Considering that these loads are applied in subsequent structural analyses, cost-intensive oversizing of pipes and their supports can be avoided.

REFERENCES

ASME B31.1 (2010). Power Piping - *ASME Code for Pressure Piping B31*. ASME, New York, NY.

ASME B31.3 (2016). Process Piping - *ASME Code for Pressure Piping B31*. ASME, New York, NY.

ASME B31.4 (1998). Pipeline Transportation Systems for Liquid Hydrocarbons and Other Liquids - *ASME Code for Pressure Piping B31*. ASME, New York, NY.

Bergant, A., Tijsseling, A.S., Vítkovský, J.P., Simpson, A.R. and Lambert, M.F. (2007). “Discrete vapour cavity model with improved timing of opening and collapse of cavities”, *Proceedings 2nd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, Timişoara, Romania, pp. 117-128.

Gale, J. and Tiselj, I. (2004). “WAHA (Water Hammer) Computer Code”, *The Practical Application of Surge Analysis for Design and Operation, 9th International Conference on Pressure Surges*, Chester, UK, Vol. 2, pp. 619-632.

Neuhaus, T., Dudlik, A. and Fahlenkamp, H. (2004). “Examination of Numerical Methods and Physical Modelling of Condensation Induced Water Hammer Including Gas Release”, *The Practical Application of Surge Analysis for Design and Operation, 9th International Conference on Pressure Surges*, Chester, UK, Vol. 2, pp. 569-580.

Neuhaus, T. and Dudlik, A. (2005). “Experiments and comparing calculations on thermohydraulic pressure surges in pipes”, *The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11)*, Popes Palace Conference Center, Avignon, France.

Neuhaus, T., Schaffrath, A. and Jerinić, D. (2009). “The Pressure Surge Computer Code DYVRO mod. 3: Modelling and Validation”, *The 13th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-13)*, Kanazawa City, Ishikawa Prefecture, Japan.

Neuhaus, T. and Schaffrath, A. (2011). “Tripartite Mass Transfer Model: Development, Implementation in DYVRO, Verification and Validation”, *The 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-14)*, Toronto, Canada.

Neuhaus, T. (2019). “Calculation of the Natural Frequencies of a Fluid-Filled Pipe System with the Coupled Codes DYVRO/ROHR2: Comparison with an Analytical Solution and Validation”, *SMiRT 25*, Charlotte, North Carolina, USA.

Tijsseling, A.S. (1996). “Fluid-Structure Interaction in Liquid-Filled Pipe Systems: A Review”, *Journal of Fluids and Structures*, 10, 109-146.

Toro, E.F. (2009). *Riemann Solvers and Numerical Methods for Fluid Dynamics*, 3rd Ed., Springer Verlag.

Vítkovský, J.P., Bergant, A., Simpson, A.R. and Lambert, M.F. (2006). “Systematic evaluation of one-dimensional unsteady friction models in simple pipelines”, *J. Hydraul. Engng.*, 132(7), 696-708.

Wylie, E.B., Streeter, V.L. and Suo, L. (1993). *Fluid Transients in Systems*, Prentice Hall, Englewood Cliffs, USA.