CAVITATION OF LNG IN ULTRASONIC FLOWMETERS - CFD MODELLING

J. Sluše and J. Geršl
Czech Metrology Institute
Okružní 31, 63800 Brno, Czech Republic
jsluse@cmi.cz
jgersl@cmi.cz

Abstract

In this paper a possible cavitation of LNG in ultrasonic flowmeters is investigated. The cavitation is an unwanted effect which can occur especially near the ultrasound transducer mountings where sharp edges at the wall can appear. By means of numerical modelling we predict pressure drops which can occur near the pockets in front of the ultrasound transducers. We then calculate a minimal sub-cooling of LNG below the bubble point temperature for which the cavitation can be avoided for various velocities and pressures in the fluid.

Introduction

Growing trade with LNG led to increased interest in measurement methods used for determining the LNG amount and composition. A research project “Metrology for LNG” was started within European Metrology Research Programme (EMRP) to deal with new challenges in this field. The work presented in this paper is also a part of this project.
In this paper we concern certain aspects of ultrasonic flow metering which is often used in LNG applications. Since LNG is usually quite near to its boiling point it is always necessary to consider if a regasification (cavitation) can appear during a measurement process and under which conditions it can happen. Appearance of gas in the liquid LNG can disturb a measurement and increase the measurement uncertainty significantly. In case of ultrasonic flowmeters a potential danger of cavitation can occur due to pressure drops in cavities of the meters where the ultrasonic transducers are mounted.
The aim of this work is to investigate under which circumstances (speeds, pressures) a cavitation near the transducer mounting can occur for certain geometries of the mounting. The method used is a computational fluid dynamic (CFD) simulation with OpenFoam software.

CFD modelling

Geometry of the ultrasonic meter cavities

A scheme of a pair of ultrasound transducers mounted in an ultrasonic flowmeter is depicted in Fig. 1. The cavities appear as a spaces between the transducer windows and the main pipe of the flowmeter.

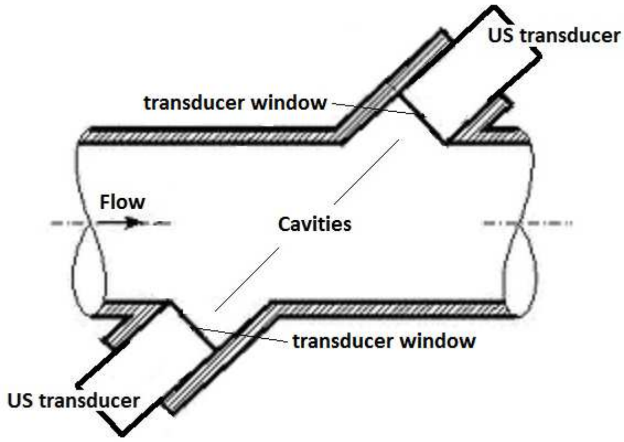

Fig. 1 Scheme of a pair of ultrasound transducers mounted in an ultrasonic flowmeter

In this paper we consider an ultrasonic flowmeter with five channels, i.e. with five transducer pairs distributed over the main pipe diameter. The shapes of the corresponding cavities therefore differ for pairs at various positions. The exact geometry was consulted with one of the producers of ultrasonic flowmeters for LNG – the Krohne company – and corresponds to their *Altosonic V LNG* meter. However the results obtained here should be valid also for other meters with similar geometries.
In our investigation we consider a meter with 100 mm diameter of the main pipe. The transducers have a cylindrical shape with diameter of 18 mm. The axis of the transducers is 45° inclined with respect to the axis of the main pipe. There are five transducer pairs with axes in the following distances from the axis of the main pipe: 0 mm, ± 25 mm (half radius) and ± 40 mm (0.8 radius). The pairs with 0 mm and ± 40 mm distance lies in the same plane and are parallel. The pairs with ± 25 mm distance lies in a plane which is perpendicular to the former one and are parallel too. Cavity edges (i.e. lines at a wall where the cavities end and the main pipe starts) are not sharp but they are rounded with a curvature radius of 0.2 mm. The transducer window is not entering the main pipe and its distance from the edge is 0.5 mm (the 0.5 mm distance does not come from a construction of the meter but from a behaviour of software for geometry creation for which 0 mm could be a problem).
The resulting cavity shapes can be seen in Fig. 2.

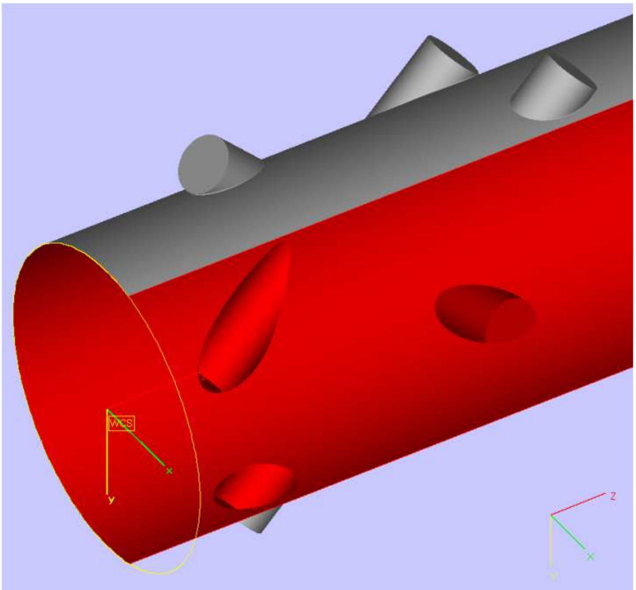

Fig. 2 Shapes of the cavities.

The software and hardware used for numerical simulations

Drawings were created in Creo Elements Direct Modeling Express 4.0. This software is free and can be downloaded from http://www.ptc.com/products/creo-elements-direct/modeling-express/. This software enables to export the geometry in STL format and to set various parameters of the STL conversion in order to achieve a required quality of the geometry. The STL file can be used for mesh generation with snappyHexMesh utility within OpenFoam sofware. Definition of boundary patches in the STL file is not available in Creo Elements and has to be done by hand – by sorting the triangles belonging to various boundary patches into corresponding groups.
For the CFD simulations OpenFoam software was used. It is a free opensource software for 3D numerical analysis of fluid flow problems. It contains solvers for Navier Stokes equations coupled to equations of variety of turbulence models. The flow of gas or liquid can be solved – in laminar or turbulent regime, steady or unsteady, compressible or incompressible. The user can choose from several algorithms the one which fits best to the physical problem. OpenFoam can be downloaded at www.openfoam.com. The OpenFoam package also contains a Paraview software for postprocessing the computed data.
The computations have been done in parallel at a set of five computers connected via network. The total number of processor cores was 20 and the total RAM memory was 178 GB.

Mesh and simulation settings

For generation of a computational mesh the snappyHexMesh utility of OpenFoam was used. Since a cavitation can occur most likely due to a local pressure drops in neighbourhood of the cavity edges it is necessary to create a mesh which is fine enough in this area. Several meshes were created with various refinement near the edges. Finally the mesh was used for which a further refinement does not lead to significant change of pressure drops.
The final mesh contains approx. 16 millions of cells which are mostly hexahedral. Typical size of cells in the center of the main pipe is 5 mm, at a wall of the main pipe 0.06 mm (with 4 wall layers), in the center of cavity 0.3 mm, at the wall of cavities 0.08 mm (without wall layers) and at the edges of the cavities 0.01 mm. The values of y+ at the wall of the main pipe were 8.3 in average and 596 in maximum. At the wall of the cavities the average was 11.6 and maximum 115. The recommended value of y+ if a turbulence model with wall functions is used is below a few hundreds.
The overall mesh is depicted in Fig. 3. The Fig. 4a represents a detail of a cavity. The Fig. 4b shows a detail of the cavity edge with curvature radius of 0.2 mm.

A fully developed profile was created in a preliminary simulation with average fluid speed of 7.5 m/s. This profile was used as a boundary velocity field at the inlet to the geometry. For pressure a zero gradient condition was used at the inlet. At the outlet the zero gradient condition was used for velocity and a fixed value was chosen for pressure. At the wall zero velocity was fixed and a pressure with zero normal gradient was selected. For the turbulence quantities the wall functions were used at the walls.

Since the major component of LNG is methane (90 % - 99 %) the material properties of LNG for the simulation were adopted from the liquid methane properties at a temperature of –165 °C and a pressure of 5 bars. The density of liquid methane under these conditions is ρ=430.1 kg/m^3 and its kinematic viscosity is ν=0.288 mm^2/s (see [1]).

The numerical problem was first solved as a turbulent, stationary, incompressible flow with *simpleFoam* solver and $k - \varepsilon$ turbulence model. However it turned out that under these assumptions the residuals of the fields oscilate and the solution does not tend to a converged state. Therefore we dropped the assumption of stationarity and we solved the problem as a non-stationary one with *pisoFoam* solver. In this case the relative residuals around 8.10^{-5} were reached for pressure and around 10^{-6}, 4.10^{-6} and 10^{-7} for velocity components.

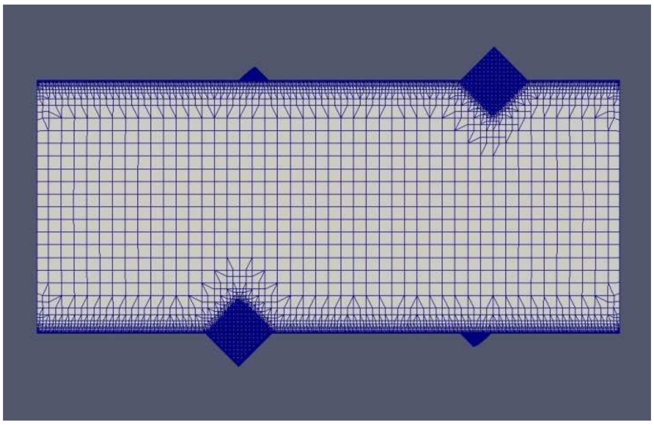

Fig. 3 The mesh – complete view.

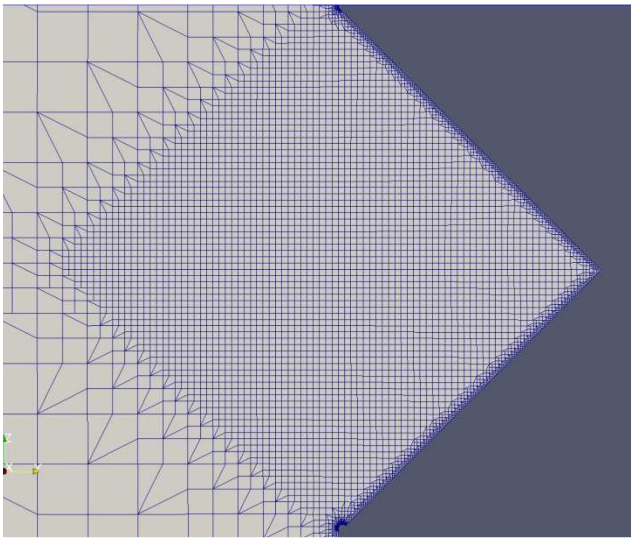

Fig. 4a Detail of a mesh in a cavity.

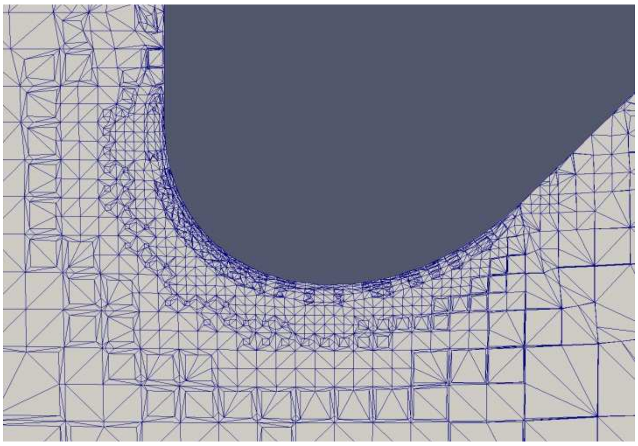

Fig. 4b Detail of a mesh near the cavity edge.

Results

The way how to investigate conditions under which cavitation can occur in the neighbourhood of cavities of ultrasonic flowmeter is to find a pressure minima in these areas. We do not simulate cavitation itself we are just interested in the maximal pressure drop in the liquid phase and circumstances under which this pressure drop can lead to regasification.

Let us denote the bulk pressure in the main pipe p and the local pressure drop Δp (we take it as a positive number). Next we denote T the actual temperature of the LNG, $T_b(p)$ the bubble point temperature at pressure p (i.e. the temperature for which a gas starts to appear) and $\Delta T_S = T_b(p) - T$ the sub-cooling temperature. The pressure drop Δp leads to a change in the bubble point temperature which is approximately equal to

$$\Delta T_b = \frac{dT_b}{dp}\Delta p. \quad (1)$$

The cavitation will occur if $\Delta T_b > \Delta T_S$.

Figs. 5 and 6 illustrate the pressure distribution near the cavities. The scale in the pictures is in density units, i.e. one unit of the scale is 430.1 Pa. From the figures we see that the pressure minima occur in very close neighbourhood of the edge. Near the edge the pressure gradient is large as the fluid has to pass the edge as we can see in the Fig. 8 with the velocity field. In Fig. 5 there is also another local pressure minimum in the center of the cavity. This minimum corresponds to a center of vertex which is depicted in Fig. 7. The value of pressure drop in this point is not so large as near the edge.

In table 1 we list a range of maximal pressure drops which occur near the particular cavities (the cavity numbering is in Fig. 9). Since the solution is not stationary the maximal pressure drop is not constant but it changes with time (that's why the range and not a number). The values obtained from computation are only that for inlet velocity of 7.5 m/s. The values for other velocities were obtained from a scaling property of pressure drops which change with square of velocity, i.e. the pressure drop for a velocity v is given as

$$\Delta p(v) = \Delta p(7.5).\left(\frac{v}{7.5}\right)^2,$$

where v is a velocity in m/s. This formula should give a good estimation if a flow topology is not changed. This formula was verified in case of stationary calculations which have been done for several velocities (however, the convergence was worse than for the non-stationary case).

The graph Fig. 10 shows the bubble curve of LNG, i.e. the curve where gas starts to appear. This curve is valid for one specific LNG composition which is quite typical (molar fractions: methane 94%, ethane 4%, propane 0.9%, butane 0.2%, nitrogen 0.9%). The data for this curve were provided by NEL. The second graph Fig. 11 shows the pressure derivative of the bubble curve. This derivative occurs in the formula (1). We can see that the derivative value is the largest for low pressures and decreases with an increasing pressure.

The tables 2 and 3 show the values of bubble temperature shift ΔT_b which appears due to the pressure drop near the cavity edges. The values were calculated from the higher value of the range of pressure drops in table 1 using the formula (1).

For the extreme case of high velocity (12.5 m/s) and low pressure (1 bar) the maximal value of the bubble temperature shift is 9.8 K. It means that if the fluid would be sub-cooled by less than 9.8 K the cavitation would occur. However, the velocities in the LNG applications are usually lower. The velocity of 12.5 m/s is an extreme case which usually does not occur in practise. For the velocity of 7.5 m/s the temperature shift value is already 3.5 K for 1 bar and 1 K for 5 bar. It means that if the velocity is below 7.5 m/s the sub-cooling by 3.5 K is sufficient to avoid cavitation for any bulk pressure higher than atmospheric pressure.

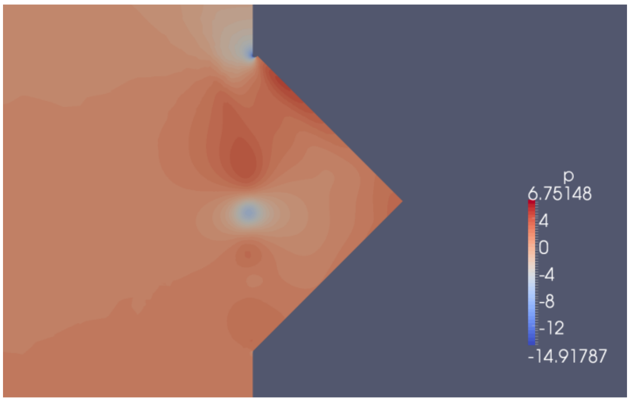

Fig. 5 Pressure distribution in one of the cavities. The unit of the scale is 430.1 Pa. The bulk pressure is 0 Pa. We can see local pressure minima near the downstream edge and in the center.

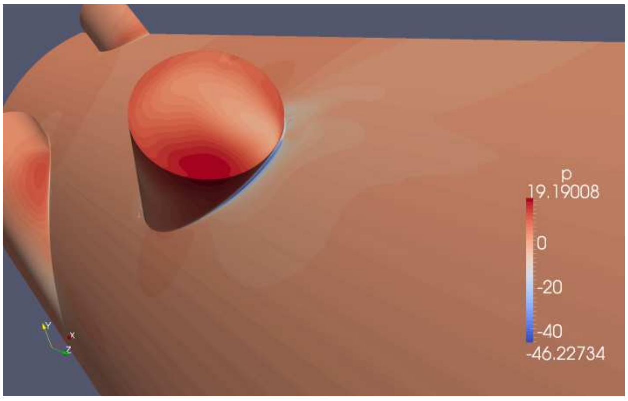

Fig. 6 Pressure distribution at the wall. The unit of the scale is 430.1 Pa. The bulk pressure is 0 Pa. We can see local pressure minimum near the downstream edge.

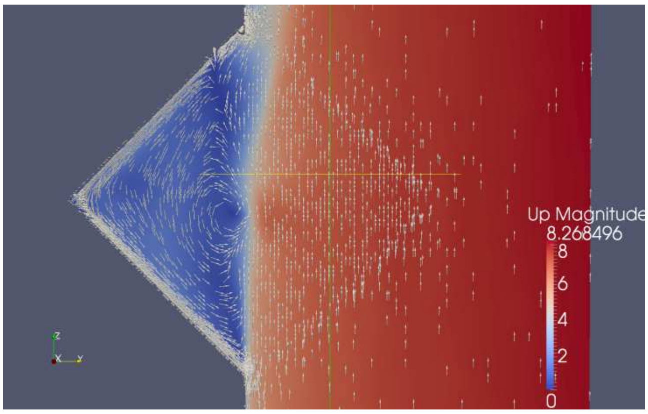

Fig. 7 Velocity field in one of the cavities. The unit of the scale is m/s. We can see a vertex in the center of the cavity.

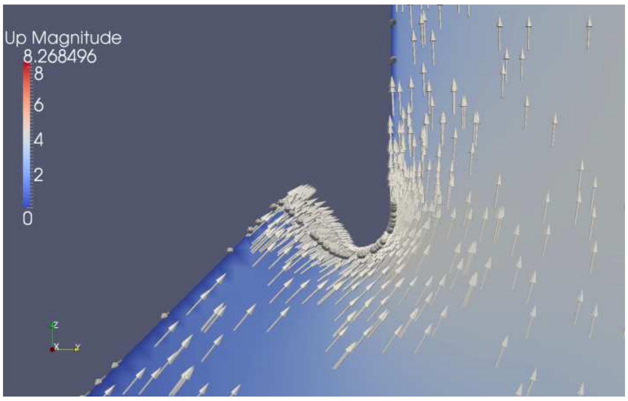

Fig. 8 Velocity field near the edge of one of the cavities. We can see how the fluid moves along the edge.

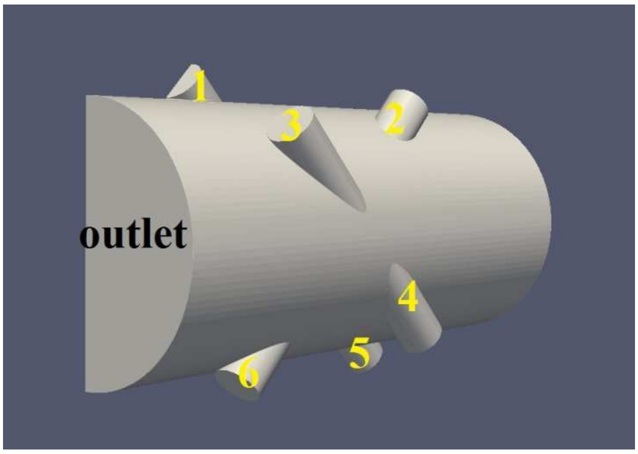

Fig. 9 Numbering of the cavities.

	Pressure drops Δp (kPa)					
cavity No.	2.5 m/s		7.5 m/s		12.5 m/s	
1	0.8	1.2	6.9	10.8	19.2	30.0
2	1.1	1.6	10.1	14.8	28.0	41.1
3	1.5	2.1	13.4	19	37.2	52.7
4	0.8	1.0	7.1	8.6	19.7	23.9
5	0.6	0.7	5.8	6.2	16.1	17.2
6	2.2	3.0	19.9	27.4	55.2	76.1

Tab. 1 Ranges of pressure drop maxima for particular cavities. The value of maximal pressure drop in a given time lies between the values in the left and the right column for a given velocity. The values for 7.5 m/s come from the CFD simulation. The other values are calculated.

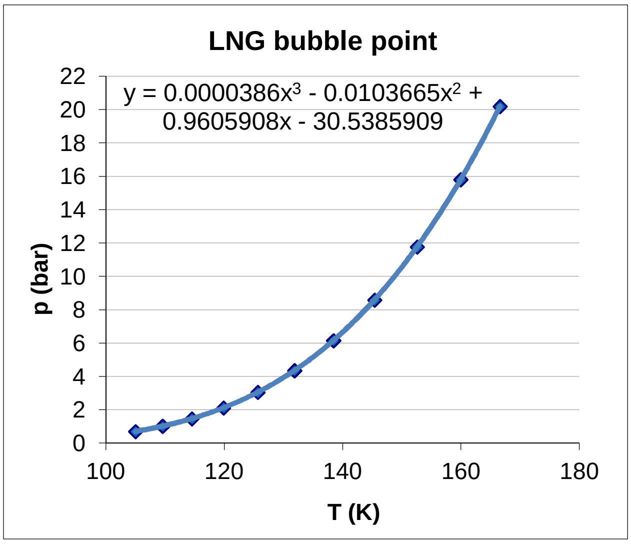

Fig. 10 Bubble point of LNG (molar fractions: methane 94%, ethane 4%, propane 0.9%, butane 0.2%, nitrogen 0.9%). Values provided by NEL. Third order polynomial fit added.

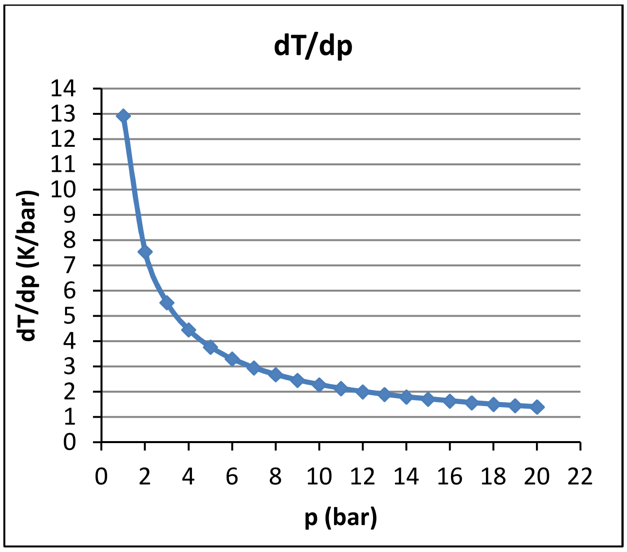

Fig. 11 Pressure derivative of the bubble point curve.

	ΔT_b (K)		
cavity No.	2.5 m/s	7.5 m/s	12.5 m/s
1	0.15	1.4	3.9
2	0.21	1.9	5.3
3	0.27	2.5	6.8
4	0.12	1.1	3.1
5	0.09	0.8	2.2
6	0.39	3.5	9.8

Tab. 2 Bubble point temperature shifts for a bulk pressure of 1 bar (dT/dp = 12.92 K/bar).

	ΔT_b (K)		
cavity No.	2.5 m/s	7.5 m/s	12.5 m/s
1	0.05	0.41	1.13
2	0.06	0.56	1.55
3	0.08	0.72	1.99
4	0.04	0.32	0.90
5	0.03	0.23	0.65
6	0.11	1.03	2.87

Tab. 3 Bubble point temperature shifts for a bulk pressure of 5 bar (dT/dp = 3.77 K/bar).

Conclusions

OpenFoam software for fluid flow simulations was used to predict pressure drops in LNG flow through ultrasonic flowmeters, especially in a neighbourhood of cavities which are between ultrasound transducer windows and the main pipe of the flowmeter. Knowledge of these pressure drops is important for determining conditions when cavitation can occur and can help to avoid the cavitation in order to keep the measurement uncertainty in a reasonable range.
The pressure drops were simulated for a geometry corresponding to a five channel ultrasonic flowmeter Krohne Altosonic V LNG. However, the results provide estimates for any similar geometry.
The largest pressure drop predicted by the simulation for a fluid velocity of 7.5 m/s was around 27 kPa. The pressure drops decrease with a decreasing fluid velocity so if we don't go above 7.5 m/s with the flow velocity (which is usually the case for LNG) the pressure drop of 27 kPa should not be exceeded.
Minimal sub-cooling (sub-cooling = difference between actual temperature and bubble point temperature at given pressure) for which a cavitation is avoided was also determined for several fluid velocities and bulk pressures in the LNG. For example for 7.5 m/s and atmospheric pressure in the main LNG stream the minimal sub-cooling is 3.5 K. The larger is the pressure in the main stream of LNG the smaller sub-cooling is needed to avoid the cavitation (e.g. for 5 bars and 7.5 m/s it is only 1 K).

Acknowledgement

The research leading to the results discussed in this paper has received funding from the European Metrology Research Programme (EMRP). The EMRP is jointly funded by the EMRP participating countries within Euramet and the European Union.

References

[1] www.fluidat.com