Calibration facility for liquefied natural gas flow-meters – CFD modeling

Jan Geršl
CMI (Czech Metrology Institute), Okružní 31, 63800 Brno, Czech Republic
jgerasl@cmi.cz

Peter Lucas
VSL (Dutch Metrology Institute), Thijsseweg 11, 2629 JA Delft, The Netherlands
plucas@vsl.nl

Oswin Kerkhof
VSL (Dutch Metrology Institute), Thijsseweg 11, 2629 JA Delft, The Netherlands
okerkhof@vsl.nl

Abstract

A new calibration facility for liquefied natural gas (LNG) flow-meters is being developed at VSL for flow-rates up to 300 m$^3$/h. The facility is based on comparing a meter under test to a set of Coriolis meters which are calibrated by a bootstrap technique using a meter traceable to an already existing laboratory-scale gravimetric LNG standard. A behavior of the LNG flow in the developed facility is predicted also by a CFD modeling using an OpenFOAM software. The modeling is focused especially to quality of LNG flow in front of the Coriolis standards including a possibility of cavitation and calculation of flow profiles. In the presentation we will show some results of this investigation – flow behavior at pipe to pipe junctions of various shapes and its impact on the flow quality downstream from the junctions. This work is done within a “Metrology for LNG” project which is a part of European Metrology Research Programme and is based on a cooperation of several European national metrology institutes.

Introduction and formulation of the CFD problem

VSL developed a gravimetric standard for LNG flow-meters which works in the range of flow-rates of (0 – 25) m$^3$/h so far. A second step is to develop and design a new calibration test rig which should raise the range at least up to 300 m$^3$/h. The method used for the up-scaling is the so called bootstrap method where one master-meter calibrated by the gravimetric method is used to calibrate other master-meters mounted in parallel and these master-meters are subsequently used to calibrate the first master-meter in a wider range. The pipe system of the bootstrap part of the up-scaled facility is schematically depicted in Fig. 1. The design of the pipe system of the bootstrap section as well as of the remaining parts of the facility was proposed by an engineering company for VSL. However, there is still an evolution in the design driven by financial and technical requirements.

In detail the bootstrap method works as follows. The master-meter A (see Fig. 1) is calibrated by the gravimetric standard in the range of (0 – 25) m$^3$/h. The six master-meters B1-B6 are calibrated by comparison to the meter A in the range of (0 – 25) m$^3$/h each of them. The meter A is then calibrated by comparison to a pair of the meters B (e.g. B1+B2) in the range of (0 - 50) m$^3$/h. Then each of the meters B1-B6 is calibrated in the range of (0 - 50) m$^3$/h by comparison to the meter A. As a result we have a set of parallel master-meters B1-B6 which work as a standard for calibrations of meters in the range of (0 – 300) m$^3$/h. All the master-meters in this facility are Coriolis type meters.

The questions that should be answered before the facility will be built are the following:

1) What will be the pressure drops near the pipe junctions? Can these pressure drops cause a regasification of the LNG?
2) What will be the flow quality (flow profile) in the pipe sections in front of the Coriolis meter standards?
3) What geometry of pipe to pipe junctions is preferred in order to achieve an optimal flow conditions in front of the Coriolis standards?
4) What will be the temperature increase caused by an ambient heat gain?

These questions were examined by means of CFD modeling using an OpenFOAM software. Three modifications of the junctions of the parallel pipes to the supplying and collecting pipes are considered in order to find an optimal velocity field in front of the Coriolis standards and minimal local pressure drops. The maximal flow-rate of 300 m$^3$/h is considered since the local pressure drops will be the largest in this case. For one of the geometries also a flow-rate of 50 m$^3$/h is investigated. A temperature increase due to a heat gain through the walls is considered as well for heat fluxes of 100 W/m$^2$ and 1000 W/m$^2$.

In the following paragraphs we will describe details of the CFD computations. First we mention how the geometries and meshes are created. Then we describe the hardware and the software used and also the OpenFOAM software settings – solvers, turbulence models, fluid properties and boundary conditions. Then we continue with an analysis of particular physical setups.

**Geometries**

The considered geometries of the pipe system are schematically depicted at Fig. 2 – Fig. 5. The system always consists of a supplying pipe with diameter 263 mm (10 inch), six parallel pipes with diameter 83 mm (3 inch) and a collecting pipe of diameter 263 mm. There is also a blind 10 inch pipe (inlet2) which corresponds to a closed connection to the master-meter A. The LNG enters the system at inlet1 and leaves at outlet. The first case considered is a geometry that contains 45° inclined junctions of the 3 inch pipes to the 10 inch pipes (see Fig. 2, 3 and 5). The edges of these junctions are sharp. Second case considered is a geometry with right angle junctions of the 3 inch pipes to the 10 inch pipes (Fig. 4). This right angle case is divided into two variants – variant with sharp edges and variant with rounded edges. The rounded edges have a radius of 10 mm in this case (see Fig. 5).
Coordinates for the 45° inclined geometry are defined in Fig. 2. The coordinate definition is analogous for all other cases.

Valves and the Coriolis meters itself were not included in the simulation geometry for sake of simplicity. Since there are the same valves and meters in each branch of the system with the same pressure drop as a function of flow-rate adding the valves and the meters would cause higher pressure drop in the branches with higher flow-rate and therefore lowering the flow-rate and equalizing the flow-rates in particular branches. Therefore the installation of the meters and the valves would lead to more equalized flow distribution than it is without the valves and the meters. The valves should be fully open during the measurements so the velocity field should not be affected significantly.

Fig. 2 Scheme of the pipe system with the 45° inclined pipe to pipe junctions; dimensions in mm; pipe numbering and boundary patches

Fig. 3 3D view of the pipe system
Fig. 4 Scheme of the pipe system with the 90° pipe to pipe junctions; dimensions in mm

Fig. 5 Detail of the 45° inclined junction (left) and of the 90° junction with rounded edges (right)

All the geometries 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/](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 the OpenFOAM. 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.

**Mesh**

Meshes for all the geometries were generated using the snappyHexMesh tool of OpenFOAM. A near wall cells thickness was set to 0.15 mm in all the cases. It corresponds to $y^+ = 75$ for inlet flow-rate of 300 m$^3$/h, LNG viscosity of 0.288 mm$^2$/s and friction factor $f = 0.012$. This value of $y^+$ is satisfactory for calculations with wall functions. The number of cells is around 16 300 000 for all the cases. Finer mesh is used near the junctions of pipe 1 and pipe 6 to the supplying and to the collecting pipe.
Junctions of pipe 1 and pipe 2 to the supplying pipe can be seen in Fig. 6. Other mesh visualization is in Fig. 7.

**Fig. 6** Junctions of pipe 1 and pipe 2 to the supplying pipe

**Fig. 7** Cut of the 10 inch supplying pipe (left) and cut of a 3 inch pipe (right).

**Computation with OpenFOAM**

The OpenFOAM software is a free open source software for 3D numerical analysis of fluid flow problems. The OpenFOAM package contains tools for mesh creation (snappyHexMesh) and for post-processing of the computed data (Paraview) as well. Other information on OpenFOAM can be found at its web page [www.openfoam.com](http://www.openfoam.com).

In order to answer the questions stated above simulations of steady, viscous, turbulent, incompressible flow with heat gain were performed. A *buoyantBoussinesqSimpleFoam* solver of OpenFOAM together with a standard *k-\( \varepsilon \)* turbulence model were used in all cases. This solver together with the *k-\( \varepsilon \)*
turbulence model solves stationary Reynolds averaged Navier-Stokes equations coupled to $k$-$\varepsilon$ turbulence equations plus a heat convection/conduction equation. The heat transfer equation is coupled to the Navier-Stokes equations through the density of the fluid. Density variations are allowed here but only as a consequence of thermal expansion and not due to pressure changes. The thermal expansion coefficient has to be set in fluid parameters. In our simulations the thermal expansion was set to zero and therefore no buoyancy effects and density variations were taken into account.

The computation was done in parallel at 20 processor cores with 52 GB RAM of memory.

Properties of the fluid

The fluid considered is a liquefied natural gas. The LNG consists from several components. The main one is methane with molar fraction exceeding 90%.

<table>
<thead>
<tr>
<th>Parameters of liquid methane, $T = -165 , ^\circ C$, $p = 5 , \text{bar}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar kinematic viscosity</td>
</tr>
<tr>
<td>$\nu=0.288 , \text{mm}^2/\text{s}$</td>
</tr>
<tr>
<td>Laminar Prandtl number</td>
</tr>
<tr>
<td>$Pr=1.808$</td>
</tr>
<tr>
<td>Turbulent Prandtl number</td>
</tr>
<tr>
<td>$Pr_t=0.85$</td>
</tr>
<tr>
<td>Density</td>
</tr>
<tr>
<td>$\rho=430.1 , \text{kg/m}^3$</td>
</tr>
<tr>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$k_t=0.2003 , \text{W/m.K}$</td>
</tr>
<tr>
<td>Thermal expansion coefficient</td>
</tr>
<tr>
<td>0 (expansion neglected)</td>
</tr>
</tbody>
</table>

Tab. 1 Parameters of the liquid methane

Boundary conditions

Boundary conditions were defined at four boundary patches – inlet1, inlet2, outlet and wall (see Fig. 3). The inlet2 boundary patch is a closed second inlet and therefore the boundary conditions for inlet2 are the same as for the patch wall (except temperature boundary conditions).

Velocity field at inlet1 is homogeneous and perpendicular to the inlet1 patch. The velocity magnitude corresponds to the flow-rates under consideration – namely the maximal flow-rate of 300 m$^3$/h (velocity 1.511 m/s) and a flow-rate of 50 m$^3$/h (velocity 0.252 m/s). The velocity is zero at the wall. At the outlet there is a zero gradient condition for the velocity field.

Ratio of pressure and density at inlet1 as well as at the wall patch satisfies a zero gradient condition. At the outlet the pressure/density is fixed to zero value. Arbitrary constant value can be added to the pressure/density since the equations contain only derivatives of pressure/density.

Turbulent kinetic energy $k$ and dissipation rate of turbulence $\varepsilon$ are given by a specific wall functions at the wall patch. These wall functions are predefined in OpenFOAM. At the outlet zero gradient condition holds and at inlet1 the $k$ and $\varepsilon$ have a fixed values which are estimated according to formulas for fully developed pipe flow.

Temperature at inlet1 is homogeneous fixed value of 108.15 K. At outlet patch the zero gradient condition holds. At the wall patch a fixed homogeneous temperature gradient is given which corresponds to certain heat flux through the wall. The heat flux $q$ and the orthogonal temperature gradient $\partial T/\partial n$ are related by a formula:

$$q = -k_t \frac{\partial T}{\partial n}$$

where $k_t$ is a thermal conductivity of the fluid. Three values of wall heat flux were investigated – 20 W/m$^2$ which corresponds to a vacuum jacket insulation of the pipes, 100 W/m$^2$ which corresponds to an insulation by a plastic foam material and 1000 W/m$^2$. The temperature gradient is set to zero at inlet2. Besides the boundary conditions for temperature certain thermal wall function have to be chosen in OpenFOAM which determine a near wall behavior of the temperature.
Velocity profiles for various pipe to pipe junction geometries

Regarding the velocity profiles in front of the positions of Coriolis mater-meters we investigate especially asymmetry of the flow and its swirl. The influence of the velocity profile to a performance of the Coriolis meters is hardly predictable. However, one can expect some influence especially in case of an asymmetric flow when the fluid velocity in front of one tube of the Coriolis meter differs from the velocity in front of the second one. Some theoretical and experimental work in this field was done e.g. by Kutin et al. [1] or Cheesewright et al. [2]. Cheesewright tested three different meters with different geometries. For two of the meters, severely asymmetric inlet velocity profiles (50% blockage immediately upstream of the meter) produced no detectable effect on the meter calibration. For the other meter there was an effect (-0.35 % to +0.2 %), which varied with the orientation of the asymmetry. The error change due to a swirl in front of the meters was below 0.25 % for all the meters.

For our considerations we define the asymmetry of flow in a pipe as a relative difference of velocities in two points “a” and “b” lying on an axis of a cut through the pipe in 25 % (point a) and 75 % (point b) distance from the edge (see Fig. 8). If we denote $U_y(a)$ the y velocity component (longitudinal component in case of one of the parallel pipes) in a point “a” and $U_y(b)$ the velocity component at a point “b” then the formula for asymmetry is $(U_y(b) - U_y(a)) / U_y(b) \times 100 \%$. For the asymmetry calculation we consider an a-b axis lying in x direction.

![Fig. 8 For explanation of $U_y$ asymmetry calculation](image)

All the velocity data are related to the case of maximal total flowrate 300 m³/h. Table 2 shows the asymmetry of flow in the pipe 1 and table 3 shows the same for pipe 6. The asymmetry is calculated for various distances from the supplying pipe and for various pipe to pipe junction geometries. From the table 2 we can see that the asymmetry is significantly lower for the 45° geometry than for the 90° geometries in case of pipe 1. For the pipe 6 the situation is different because the fluid is almost standing at the inlet to the pipe 6. From these results we see that the design with 45° inclined junctions is preferred in order to avoid large asymmetry values. Some of the profiles of $U_y$ field with contour plots are depicted in the right part of Fig. 9 – Fig. 14.

<table>
<thead>
<tr>
<th>$U_y$ asymmetry in pipe 1</th>
<th>junction geometry</th>
<th>45°</th>
<th>90° sharp</th>
<th>90° round</th>
</tr>
</thead>
<tbody>
<tr>
<td>y = 190 mm</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>y = 400 mm</td>
<td></td>
<td>-20.8 %</td>
<td>59 %</td>
<td>45 %</td>
</tr>
<tr>
<td>y = 900 mm</td>
<td></td>
<td>2.7 %</td>
<td>22 %</td>
<td>17 %</td>
</tr>
<tr>
<td>y = 1400 mm</td>
<td></td>
<td>3.3 %</td>
<td>13 %</td>
<td>11 %</td>
</tr>
<tr>
<td>y = 2400 mm</td>
<td></td>
<td>2.0 %</td>
<td>7.3 %</td>
<td>6.3 %</td>
</tr>
</tbody>
</table>

Tab. 2 Comparison of $U_y$ asymmetry in pipe 1 for various junction geometries
junction geometry & 45° & 90° sharp & 90° round \\ 
\hline
$U_y$ asymmetry in pipe 6 & \\ 
\hline
$\gamma = 190 \text{ mm}$ & 7.8 \% & 2.2 \% & \\ 
$\gamma = 400 \text{ mm}$ & -24.8 \% & 8.1 \% & 2.1 \% \\ 
$\gamma = 900 \text{ mm}$ & 1.2 \% & 3.4 \% & 0.7 \% \\ 
$\gamma = 1400 \text{ mm}$ & 0.2 \% & 2.3 \% & 1.3 \% \\ 
$\gamma = 2400 \text{ mm}$ & -1.2 \% & 0.7 \% & 0.5 \% \\
\hline

Tab. 3 Comparison of $U_y$ asymmetry in pipe 6 for various junction geometries

To quantify a swirl for certain cut through a pipe we use a maximum of a magnitude of a projection of the velocity field to the plane of the cut (i.e. a maximal transversal velocity). In case of the parallel pipes we denote this quantity $U_{pym}$ and we have

$$U_{pym} = \max \left( \sqrt{U_x^2 + U_z^2} \right)$$

for a circle given by certain cut through a pipe. In order to get an idea about significance of the transversal components we normalize this value by a typical longitudinal velocity in given pipe. As a typical longitudinal velocity a maximum of $U_y$ for $\gamma = 2400 \text{ mm}$ is taken for given pipe which we denote $U_{ym}$ (see Tab. 5). The values in table 4 are therefore calculated as $U_{pym,rel} = U_{pym}/U_{ym} \times 100 \%$.

The table 4 shows the $U_{pym,rel}$ values in pipe 1 and pipe 6 for various distances from the supplying pipe and for various pipe to pipe junction geometries. The flow pattern in transversal velocity projection can differ from case to case. The projected velocity field $U_{py}$ is depicted in the left part of the Fig. 9 – Fig. 14. We can observe four-vertex pattern (e.g. pipe 1, 45° case), two-vertex patterns (most common case) or single-vertex pattern (e.g. pipe 6, 90° round case).

From Tab. 4 we see that for pipe 1 the differences in transversal velocities are not dramatic if we compare various geometries. Larger difference is observed for the pipe 6. The reason is that the velocity at the inlet to the pipe 6 is very low and the swirls are mostly generated by the elbows behind the 45° inclined junctions and not at the connections of the 3 inch pipes to the 10 inch pipe.

| $U_{pym,rel}$ | pipe 1 & pipe 6 \\ 
| $\gamma = 190 \text{ mm}$ & 31.8 \% & 21.3 \% & 7.7 \% & 3.1 \% \\ 
| $\gamma = 400 \text{ mm}$ & 14.7 \% & 10.4 \% & 9.4 \% & 17.0 \% & 1.24 \% & 0.51 \% \\ 
| $\gamma = 900 \text{ mm}$ & 1.81 \% & 2.18 \% & 3.0 \% & 2.5 \% & 0.20 \% & 0.12 \% \\ 
| $\gamma = 1400 \text{ mm}$ & 0.68 \% & 0.86 \% & 1.46 \% & 1.29 \% & 0.078 \% & 0.076 \% \\ 
| $\gamma = 2400 \text{ mm}$ & 0.24 \% & 0.27 \% & 0.56 \% & 0.55 \% & 0.027 \% & 0.049 \%

Tab. 4 Comparison of $U_{pym,rel}$ for various geometries

| $U_{ym}$ (m/s) | Pipe No. | 1 | 2 | 3 | 4 | 5 | 6 \\ 
| $y = 2400 \text{ mm}$ & 45° & 3.182 & 3.022 & 2.878 & 2.773 & 2.687 & 2.608 \\ 
| & 90° sharp & 2.616 & 2.647 & 2.794 & 2.903 & 3.002 & 3.094 \\ 
| & 90° round & 2.670 & 2.681 & 2.797 & 2.876 & 2.946 & 3.022

Tab. 5 Distribution of $U_{ym}$ in the six parallel pipes for various geometries
Fig. 9 Cut y = 1400 mm - pipe1; inlet flowrate 300 m$^3$/h; 45° inclined junctions; values in mm/s; contour step 100 mm/s

Fig. 10 Cut y = 1400 mm - pipe6; inlet flowrate 300 m$^3$/h; 45° inclined junctions; values in mm/s; contour step 100 mm/s

Fig. 11 Cut y = 1400 mm - pipe1; inlet flowrate 300 m$^3$/h; 90° sharp junctions; values in mm/s; contour step 100 mm/s
Fig. 12 Cut y = 1400 mm - pipe 6; inlet flowrate 300 m³/h; 90° sharp junctions; values in mm/s; contour step 100 mm/s

Fig. 13 Cut y = 1400 mm - pipe 1; inlet flowrate 300 m³/h; 90° round junction; values in mm/s; contour step 100 mm/s

Fig. 14 Cut y = 1400 mm - pipe 6; inlet flowrate 300 m³/h; 90° round junction; values in mm/s; contour step 100 mm/s
Local pressure drops for various pipe to pipe junction geometries

Local pressure drops are a potential cause of LNG regasification which leads to disturbed flow and can influence the Coriolis master-meters performance. The pressure drops should be minimized in order to avoid a gas in the flow and flow disturbances. Therefore these pressure drops which occur especially at the junctions of the parallel pipes to the supplying pipe were investigated for various junction geometries. The results were compared in order to find an optimal design.

The pressure drops for an inlet flowrate of 300 m$^3$/h are summarized in table 6. All the pressure drops are related to the average inlet value of pressure, i.e. the values in the table are calculated as $p_{ai} - p(x)$ where $p(x)$ is an absolute pressure at a point x and $p_{ai}$ is the average pressure at the inlet1 patch. The “max pipe1 pressure drop” in the table is a maximal pressure drop at the junction of pipe 1 to the supplying pipe. The “max upstream pressure drop” is the maximal value of pressure drop in front of the Coriolis standards position. The “outlet pressure drop” is a pressure drop at the outlet patch. The maximal pressure drop upstream the standards is always near an edge (or rounded edge) of a junction of one of the parallel pipes to the supplying pipe. In case of the 90° junctions the lowest pressure occurs at the upstream part of the edge. See Fig. 15 for pressure distribution in the case with sharp 90° junctions and also Fig. 16 for a flow pattern in this case. In case of the 45° inclined junction the lowest pressure occurs at the opposite side of the edge – at its sharpest part.

In the table 6 we show not only the maximal upstream pressure drops but also the values of maximal pressure drop for pipe 1 in order to compare pressure drops at one location for different geometries.

<table>
<thead>
<tr>
<th>geometry</th>
<th>45° max pipe1 pressure drop</th>
<th>90° sharp max pipe1 pressure drop</th>
<th>90° round max pipe1 pressure drop</th>
</tr>
</thead>
<tbody>
<tr>
<td>max upstream</td>
<td>6897 Pa</td>
<td>7325 Pa</td>
<td>5717 Pa</td>
</tr>
<tr>
<td>pressure drop</td>
<td>6897 Pa</td>
<td>8394 Pa</td>
<td>6026 Pa</td>
</tr>
<tr>
<td>outlet pressure</td>
<td>2998 Pa</td>
<td>3620 Pa</td>
<td>3101 Pa</td>
</tr>
</tbody>
</table>

Tab. 6 Comparison of pressure drops for various junction geometries

The best results (smallest pressure drops) are achieved for the rounded 90° geometry but the differences between various geometries are not very significant – e.g. the rounding of the sharp 90° edge of the junction of pipe 1 causes a decrease of the pressure drop by 22%.

Fig. 15 Pressure/density ratio at a junction of pipe 4 to the supplying pipe (z = 36 mm); the sharp 90° junction; inlet flowrate 300 m$^3$/h; values in mm$^3$/s$^2$
Extrapolation of pressure drops to higher velocities

The pressure drops listed in Tab. 6 are for the inlet flow rate of 300 m$^3$/h flowing through the 10 inch supplying pipe. During the simulations the concept of the test rig was little bit redesigned from some technical reasons. In the new design the maximal flow-rate is 400 m$^3$/h instead of 300 m$^3$/h and the supplying pipe has 6 inch diameter instead of 10 inch. Therefore the inlet velocity increases by a factor of 3.7 for the new design. If the pressure drop scales with $v^2$ then we obtain an increase in pressure drops by a factor of 13.7. Therefore we should multiply the values in Tab. 6 by 13.7 in order to get an estimate for the new test rig design if we suppose that the flow topology remains the same. Since not only the simulations for 300 m$^3$/h were done but also a simulation for 50 m$^3$/h in case of the geometry with 45° inclined junctions, the pressure drop scaling can be verified by comparing these two setups. The velocity increase factor is 6 in this case and the pressure drop increase factor should be 36. For the case with the inlet flow rate of 50 m$^3$/h the calculated maximal pressure drop at the junction of pipe 1 is 196.5 Pa. This leads to pressure drop ratio of 35.1 which is in a good agreement with the assumption.

Translation of pressure drops to danger of boiling

For certain pressure, temperature and composition of LNG we can define a difference of actual temperature and a temperature of bubble point for given pressure $\Delta T_{bp}(T,p) = T_{bp}(p)-T$. The bubble point is a thermodynamic situation when bubbles of gas starts to appear in the liquid. The $\Delta T_{bp}(T,p)$ quantity therefore expresses how far we are from re-gasification in terms of temperature. Pressure drop from $p_1$ to $p_2$ leads to a change $\delta T_{bp}(p_1,p_2) = \Delta T_{bp}(T,p_1) - \Delta T_{bp}(T,p_2) = T_{bp}(p_1) - T_{bp}(p_2) \approx dT_{bp}(p)/dp \cdot (p_1-p_2)$.

The curve $T_{bp}(p)$ (or an inverse curve respectively) is depicted in Fig. 17 for a pressure range up to 20 bars. Slope of the curve is: $dT_{bp}(p)/dp = 7.55$ K/bar for pressure of 2 bars and $dT_{bp}(p)/dp = 2.00$ K/bar for pressure of 12 bars. The values 2 bars and 12 bars are chosen since they are near the minimal and maximal pressures planned for the test rig. Table 7 shows local pressure drops at junction of the pipe 1 to the supplying pipe for the computed cases (old design) and also the recalculated values for the new design (multiplying by 13.7). The corresponding bubble temperature shifts for two pressures are listed too.
y = 0.0000386x^3 - 0.0103665x^2 + 0.9605908x - 30.5385909

Fig. 17 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.

<table>
<thead>
<tr>
<th></th>
<th>geometry:</th>
<th>45°</th>
<th>90° sharp</th>
<th>90° round</th>
</tr>
</thead>
<tbody>
<tr>
<td>old design</td>
<td>max. pressure drop</td>
<td>0.069 bar</td>
<td>0.073 bar</td>
<td>0.057 bar</td>
</tr>
<tr>
<td></td>
<td>δT_{bp} (p_1 = 2 bar)</td>
<td>0.52 K</td>
<td>0.55 K</td>
<td>0.43 K</td>
</tr>
<tr>
<td></td>
<td>δT_{bp} (p_1 = 12 bar)</td>
<td>0.14 K</td>
<td>0.15 K</td>
<td>0.11 K</td>
</tr>
<tr>
<td>new design</td>
<td>max. pressure drop</td>
<td>0.95 bar</td>
<td>1.00 bar</td>
<td>0.78 bar</td>
</tr>
<tr>
<td></td>
<td>δT_{bp} (p_1 = 2 bar)</td>
<td>7.2 K</td>
<td>7.6 K</td>
<td>5.9 K</td>
</tr>
<tr>
<td></td>
<td>δT_{bp} (p_1 = 12 bar)</td>
<td>1.9 K</td>
<td>2.0 K</td>
<td>1.6 K</td>
</tr>
</tbody>
</table>

Tab. 7 Pressure drops at junction of pipe 1 to the supplying pipe for the old and the new design and the corresponding temperature shifts

The values in the table 7 means that, e.g. if we have the new design and a maximal flowrate with inlet pressure of 2 bars, a cavitation will occur if the fluid is not subcooled more than 7.2 K (in case of 45° inclined junction) below the bubble point for 2 bars.

Temperature increase for various insulations and flow regimes

Other possible cause for regasification of LNG is a temperature increase due to a heat gain through the pipe walls. As we already told before the buoyantBoussinesqSimpleFoam solver solves also a heat transfer equation with given heat flux through the wall. A temperature increase was calculated for various heat flux values which correspond to various degrees of insulation as was discussed before. Two inlet flowrates 300 m³/h and 50 m³/h were considered. The maximal temperature increase occurs in areas where the fluid is almost standing – i.e. in the blind parts of the pipes.

Table 8 shows a maximal temperature increase in the blind end of the supplying pipe for various values of wall heat flux and of flow rate. ∆T denotes the temperature increase with respect to the inlet
temperature (108.15 K). An example of temperature distribution in the blind end of the supplying pipe for the inlet flowrate of 50 m³/h and wall heat flux of 1000 W/m² is shown in Fig. 18.

<table>
<thead>
<tr>
<th>wall heat flux (W/m²)</th>
<th>inlet flow-rate (m³/h)</th>
<th>∆T (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>300</td>
<td>0.03</td>
</tr>
<tr>
<td>100</td>
<td>300</td>
<td>0.19</td>
</tr>
<tr>
<td>100</td>
<td>50</td>
<td>1.03</td>
</tr>
<tr>
<td>1000</td>
<td>50</td>
<td>9.98</td>
</tr>
</tbody>
</table>

Tab. 8 Temperature increase in the blind end of the supplying pipe.

From the table 8 we see that for a heat flux 100 W/m² which corresponds to an ordinary insulation by a plastic foam materials and for a flowrate of 50 m³/h which is quite near to the lower bound of the test rig (which is 25 m³/h) the temperature increase in the blind end of the supplying pipe is around 1 K. This is well covered by the planned subcooling below a bubble point.

Conclusions

LNG flow through a bootstrap part of a calibration facility was analyzed by means of CFD using the OpenFOAM software. Three types of junctions of the small (3 inch) parallel pipes to the supplying pipe were investigated in order to find an optimal design with lowest flow disturbances in front of the Coriolis master-meters. Namely 45° inclined junctions, 90° junctions with sharp edges and 90° junctions with rounded edges were considered. The results of CFD modeling showed that the lowest asymmetry in the flow is achieved for the 45° inclined junctions and the lowest local pressure drops are achieved for the geometry with rounded edges. From this point of view probably a geometry with 45° inclined junctions and rounded edges would be optimal in order to avoid cavitation and error shifts of the Coriolis standards due to the flow disturbances.

A shift in “temperature distance” from a bubble point due to local pressure drops was estimated for the original test rig design as well as for a new design which was suggested in the meantime. The shifts towards the bubble point temperature are larger then 5.9 K for all the geometries for the new design at the maximal flowrate.

A temperature increase due to a heat gain through the pipe wall was investigated too. The temperature increase was found not to be so significant if an ordinary plastic foam insulation is applied. E.g. a temperature increase in one of the blind parts of the pipe system is around 1 K for a flowrate of 50 m³/h and wall heat flux of 100 W/m².
Acknowledgements

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
